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Preface

The present script includes an introduction of the main features in the 3D design software package Catia V5. Beside the basic tools of 3D design, a number of exercises and examples point to different construction strategies in several applications. In addition to the primary functions, methods for the generation of solid components and assemblings are explained and executed by use of different examples.

Training targets:

- Sketch mode
- Basic part design
- Enhanced features of part design
- Assembly design and product structure
- Generating drawings

The script is based on Catia V5 Release 15 and will be updated continuously. To keep the paper up to date and to fulfill the requirements on the Catia V5 education at a high level, questions, critics and new inputs are sincerely welcome. Please write an email to:

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1 Introduction

The 3D CAD system CATIA V5 was introduced in 1999 by Dassault Systems. Replacing CATIA V4, it represented a completely new design tool showing fundamental differences to its predecessor.
The user interface, now featuring MS Windows layout, allows an easy integration of common software packages such as MS Office, several graphic programs or SAP-R3 products (depending on the IT environment) and others.

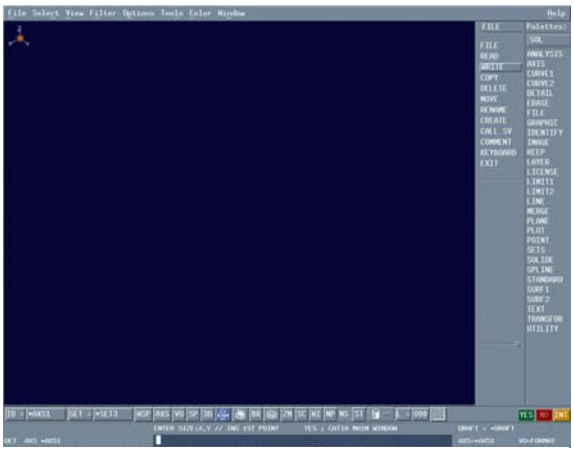


Figure 1: User interface CATIA V4.2.2

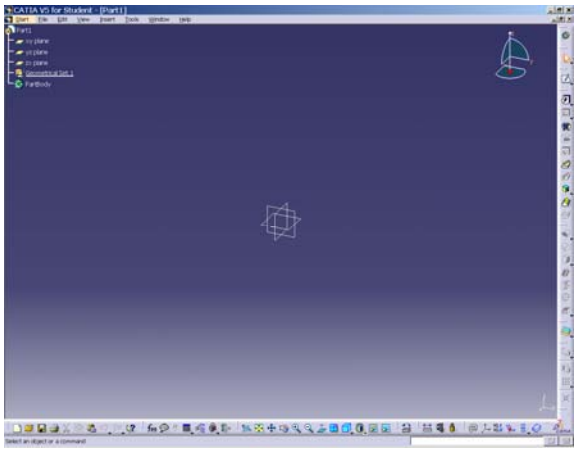


Figure 2: User interface CATIA V5 R15

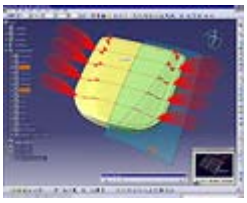
Figure 3: Selection of Workbenches

The concept of CATIA V5 is to digitally include the complete process of product development, comprising the first draft, the design, the layout and at last the production and the assembly.
The present training includes a selection of functionalities in the workbench *Mechanical Design*.

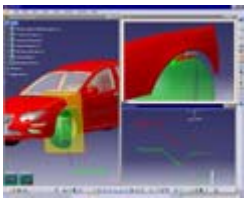
Sets of workbenches can be composed according to the user's preferences. Therefore Dassault Systems offers three different software installation versions.
The platform P1 contains the basic features and is used for training courses or for reduced functionalities. For process orientated work the platform P2 is the appropriate one. It enables, apart from the basic design features, analysis tools and production related functions. P3 comprises specific advanced scopes such as the implementation of external software packages.

1.1. An excerpt of available workbenches

Mechanical Design:
Sketches, 3D Design, 2D Drawings

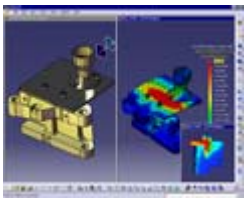


Shape:
Surface based design, Free formed surfaces

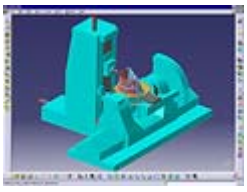


Digital Mockup
Digital Mockup, Packaging and Assembly Simulation

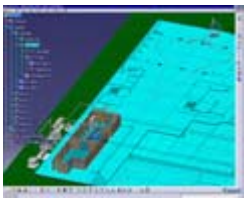
Equipment and Systems:
Integration of complex elements and components such as wiring harnesses, hydraulic systems etc.



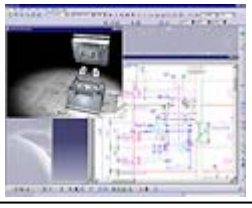
Analysis & Simulation:
Calculation tool for the design accompanying simulation and analysis



Machining:
Manufacturing simulation and control tool for numerically controlled machines



AEC Plant:
Manufacturing and production planning, Optimization of production lines



Infrastructure:
Interfaces, Comprehensive work with other software packages, Data transfer

2 The user interface of CATIA V5

Compared to CATIA V4, the desktop design is completely new. Established elements of other software packages have been integrated and several well known features can be used in CATIA V5.

Thus, figures can be directly inserted into MS Word documents out of CATIA V5, and MS Excel tables can be easily used as design tables in CATIA V5.

2.1. Graphic display

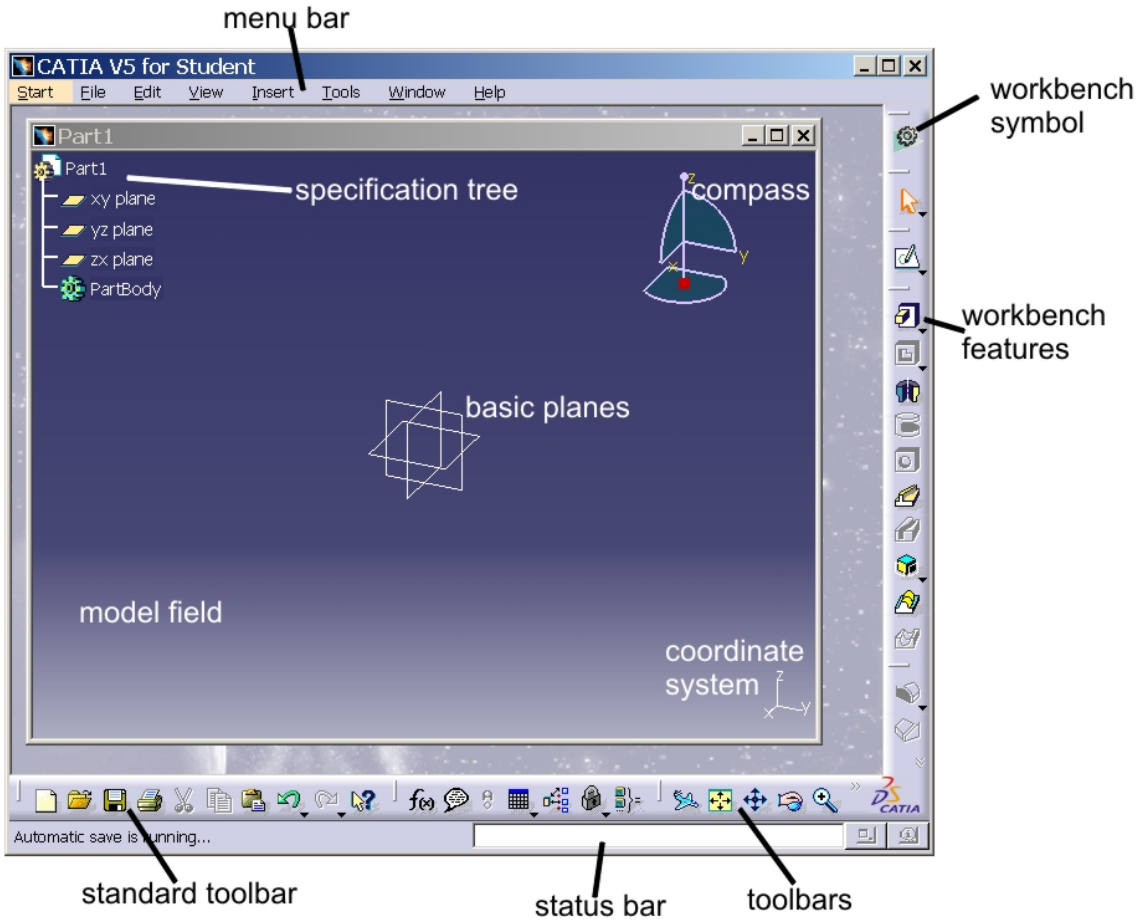


Figure 4: Graphic display in CATIA V5

- Menu bar with pull down menus for the access of CATIA features
- Workbench symbol for quick switching between the workbenches
- Standard toolbar containing common features such as *Open, Close, Print, Cut* and *Paste*
- The open window contains the model field and the specification tree
- Compass, used for changing the view and moving objects
- Status bar with instructions and prompts
- The workbench toolbar displays all the features, that can be used within a specific workbench

2.2. Mouse button assignment

A three buttoned mouse is needed to control the movements of the elements and the zooming, in the 3D-space as well as in the specification tree. The following mouse button operation is used in the default configuration of CATIA V5.

Elements in the 3D-design space:

Move ... Press and hold the middle mouse button and move the mouse.

Rotate ... Press and hold the middle mouse button. While still holding it, press and hold the left (or right) button and move the mouse.

Zooming ... Press and hold the middle mouse button. While still holding it, press the left (or right) button once and move the mouse.

Changing the center ... Click the middle mouse button on the location of the element that shall be moved to the center of the window. The window center also represents the rotation center.

Specification Tree:

Move ... Press and hold the left mouse button while the mouse points at a branch of the tree, and move the mouse.

Zooming ... Click once on a branch of the specification tree (or on the coordinate system in the right lower corner of the working space) with the left mouse button. The construction elements in the 3D space get darker, the tree is now set active. The zooming of the tree can be done as described above. Another click on a branch deactivates this function.

2.3. User defined settings

The pull down menu *Tools / Options* offers several user definable settings. For instance, display settings, construction facilities, file import and export settings, memory settings and many more settings can be adjusted. By using the *Reset* button, all parameter values can be set to the initial values fixed by Dassault Systems.

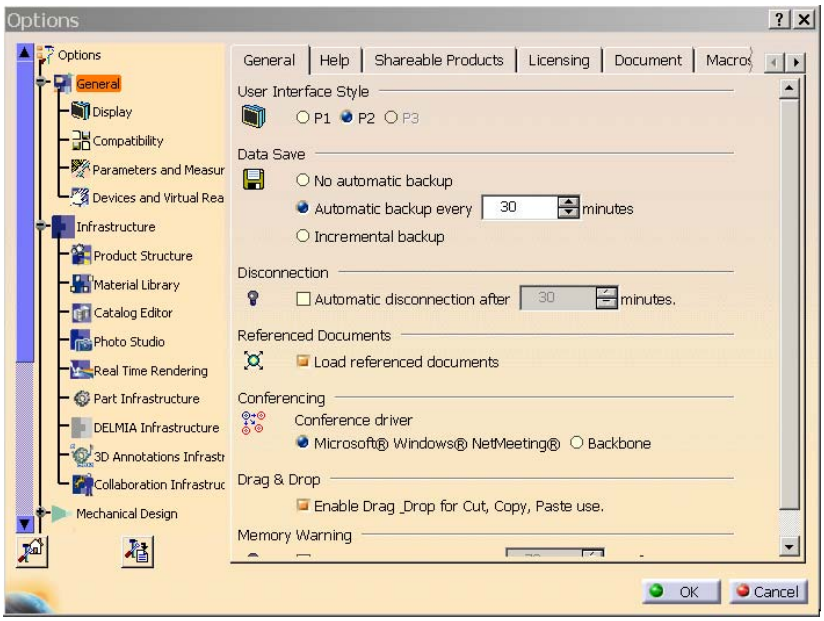


Figure 5: *Options*

3 An excerpt of menus

This chapter explains a selection of the most important menu bars of the workbench *Part Design*. A couple of basic features (*Start, File, Edit* etc.) are also available in other workbenches, other menu bars differ depending on the specific demands of the modes. The following chapters give a deeper understanding of single menu bars in different workbenches.

3.1 Start

The *Start* menu contains the workbenches defined previously. The pull down menu is used to switch from one workspace to the other. Additionally, the recently opened, the active and the previous open file names are shown. By clicking on the names, the files can be activated.

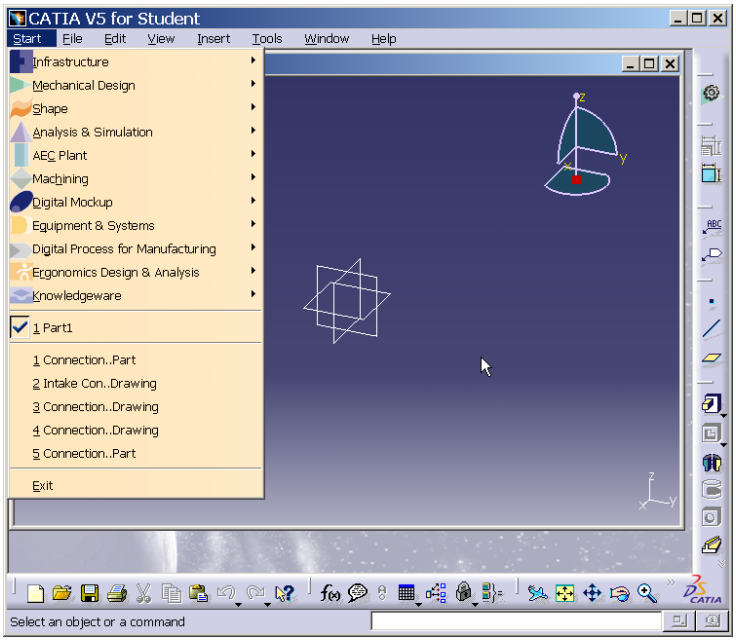


Figure 6: *Start* menu

3.2 File

File comprises all the administrative functions for opening, saving or printing files. In addition, the recently used files are displayed.

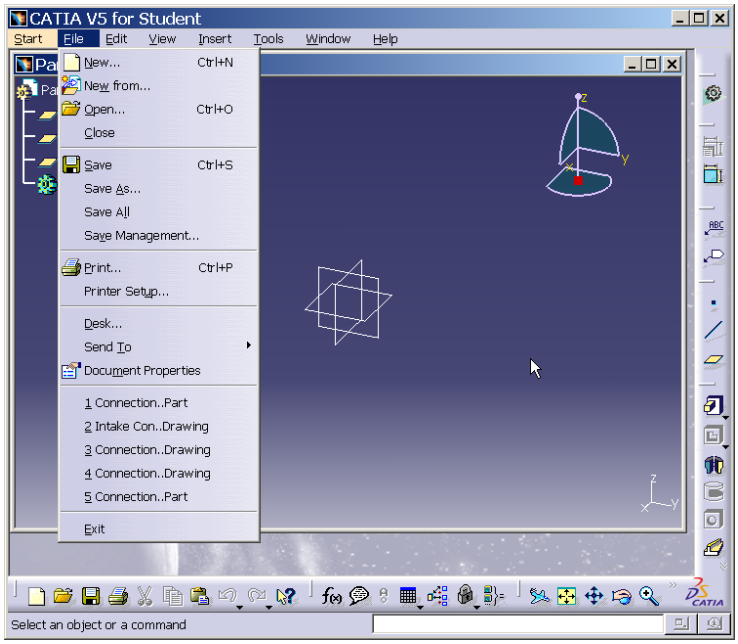


Figure 7: *File* menu

3.3 Edit

Some functions frequently needed during the design process such as *Copy, Paste, Cut* or *Delete* can be found in the menu *Edit*.

The feature *Update* is used to refresh the construction. *Undo* and *Repeat* are very useful commands to move one design step backwards or forwards again.

Search can find elements within the active document.

The commands *Selection Sets, Selection Sets Edition* and *Find Owning Selection Sets* enable the definition and the recall of selection criteria.

To edit document connections, *Links* is used.

The definition or changing of component properties happens through *Properties*.

Scan or Define In Work Object makes the navigation between elements and the definition of *In Work*-objects possible. The following construction steps are executed on this (defined) object.

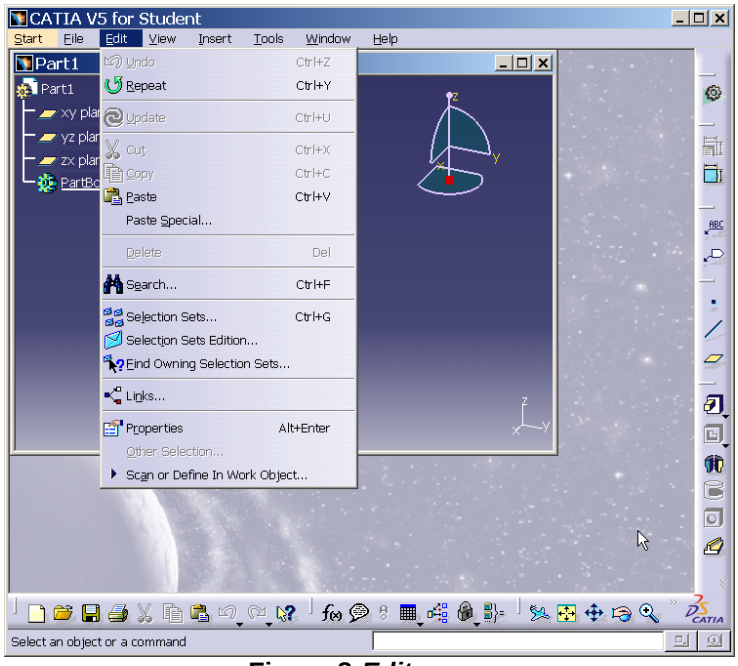


Figure 8:*Edit* menu

3.4 View

The menu *Toolbars* allows the configuration of the toolbar visualisation on the screen. By clicking on a single toolbar name, the respective toolbar can be activated or deactivated.

The *Commands List* is used to directly access commands.

The commands *Geometry, Specifications, Compass* and *Reset Compass* activate or deactivate the corresponding elements.

Tree Expansion permits the activation of the desired levels of the specification tree.

Specifications Overview and *Geometry Overview* provide an overlook of the active *Specification Tree* and geometry.

The visualization on the screen can be controlled by *Fit All In, Zoom Area, Zoom In Out, Pan* and *Rotate* with *Modify* providing even more options.

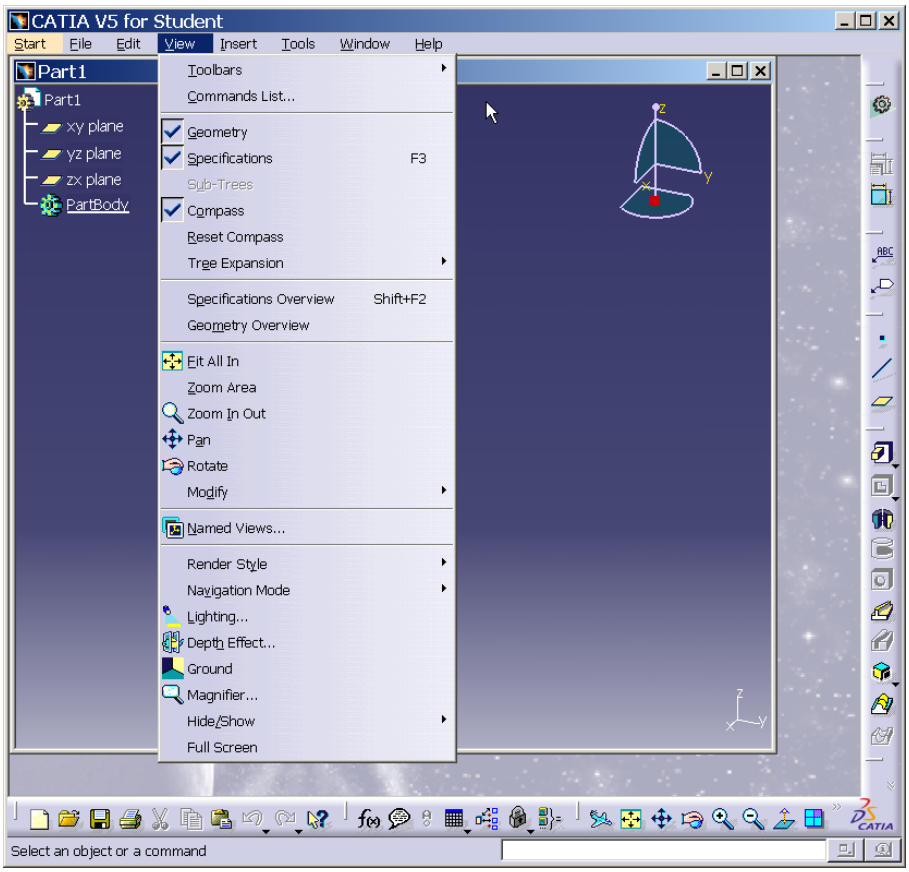


Figure 9: *View* menu

If different predefined views should be created with the possibility to quickly switch between them, the command *Named Views* can be useful.

Render Style enables the adjustment of visualization settings. Apart from standard settings, user defined render styles can be configured.

The menu *Navigation Mode* is used to choose from different types of part movement on the screen:

Fly: => Translative and rotatory movement
Walk: => Translative movement within an predefined plane

The features *Lighting* and *Depth Effect* affect the display style of shaded objects.

A base plane can be inserted via *Ground.*

Magnifier can be used to display details.

Hide/Show switches to the invisible space. Components, that are not needed at present, can be deposited in the invisible space.

To enlarge the window to its full size, *Full Screen* has to be applied.

3.5 Insert

The *Insert* menu contains specific commands for each workbench. Most of these features can be activated via the toolbars as well. A detailed description of the main commands is carried out in the specific modes *Part Design* and *Drafting*.

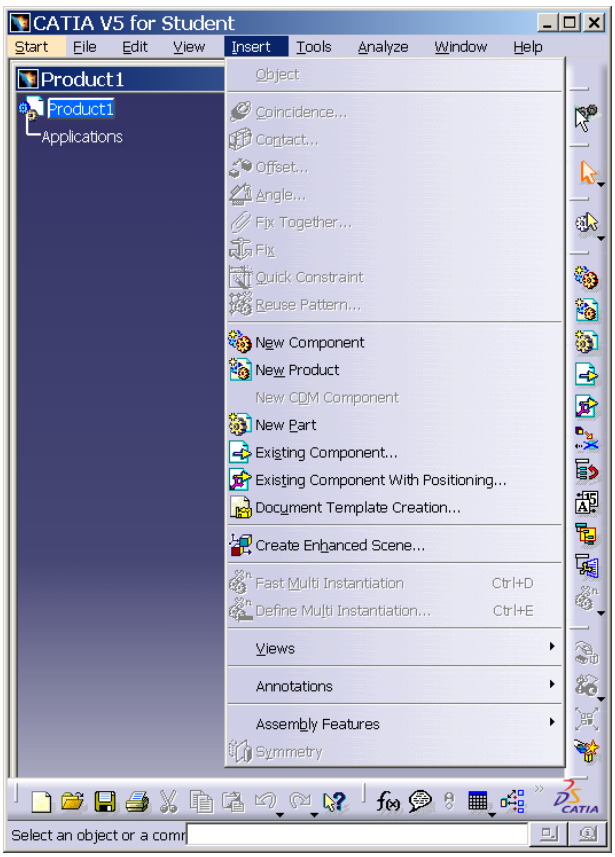


Figure 10: Menu *Insert* within the *Product* mode

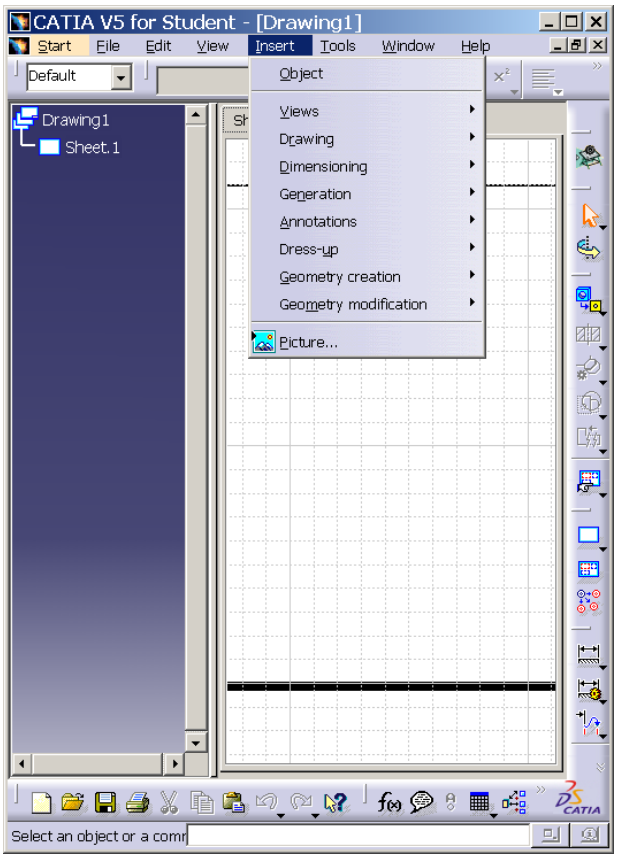


Figure 11: Menu *Insert* within the *Drafting* mode

3.6 Tools

The features contained in *Tools* control the settings and user defined features. Additionally, several workbench specific tools are available.

Formula ... The parameters of the applied operations are displayed in a window. In addition, modifications and specific applications can be defined.

Image ... Creation of pictures and videos

Macro ... The creation of macros is carried out in Visual Basic. An administration function supports a creation and organisation of libraries.

Customize ... The menu *Customize* enables user specific modifications, as there are the arrangement of menu bars or a setting of the interface language.

Visualization filters ... Layers (e.g. design spaces) can be switched visible / invisible.

Options ... Basic settings are adjusted via the *Options* - menu:

Specification tree
Navigation
Performances
Visualization
Thickness & Font
Linetype

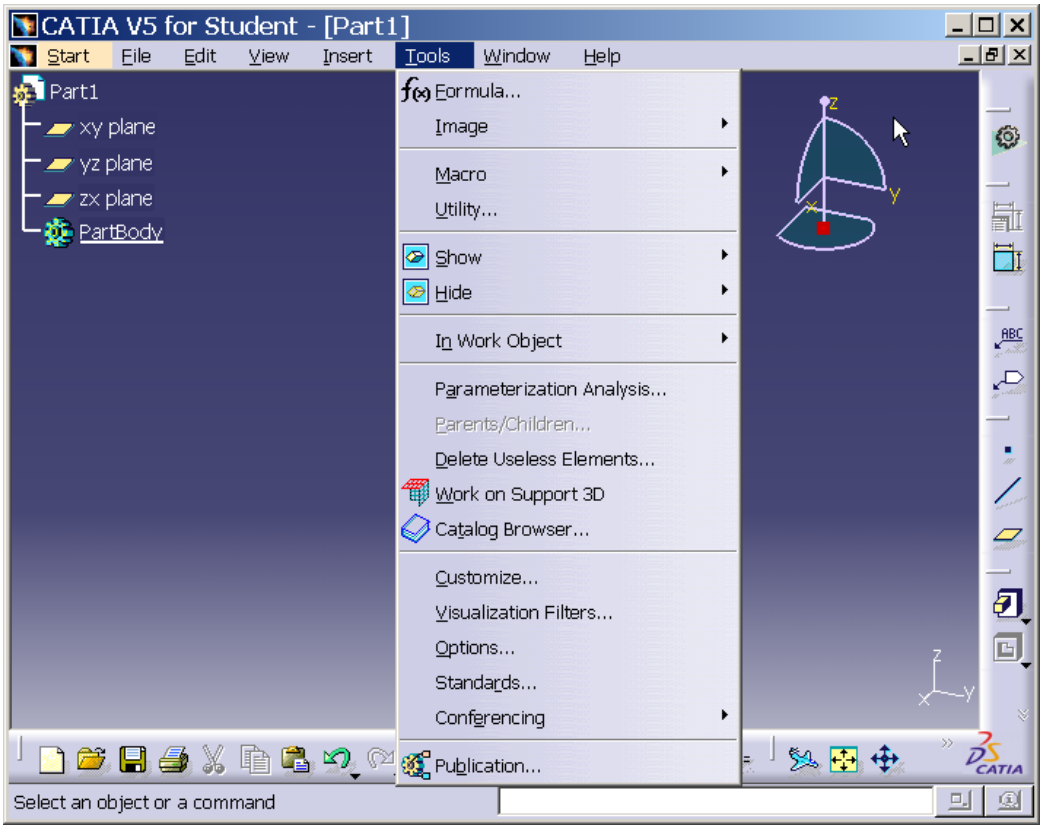


Figure 12: *Tools* menu

Standards ... To set default values for element properties, use *Standards*.

Conferencing ... *Conferencing* is needed to organize conferences.

3.7 Window

Opened windows can be arranged and new windows can be opened with the *Window* menu. Furthermore the open files are displayed there.

3.8 Help

A contextual help (*What's This?*), explaining the commands instantly and a help menu (*CATIA V5 Help*) which requires special installation, are provided by CATIA V5.

4 Toolbars in the workbench Part Design

The desired toolbars can be shown and removed using the menu *View / Toolbars*. Depending on the activated workspace, specific toolbars are available.

Beside the general toolbars

Standard
Knowledge
Workbench
Graphic Properties and
View,

some workbench specific toolbars will be explained. After switching to another workbench, the menu *Toolbars* automatically activates the accordant functions. Operational functions are not only accesseble in the according toolbars, they can also be accessed by the pull down menu *Insert*.

4.1 Standard toolbar

New ...	Creates a new part, assembly or drawing document
Open ...	Opens an existing document
Save ...	Saves the active document
Print ...	Prints the active document on the default printer, using the default printer settings
Cut ...	Removes the selection from the active document and places it on the clipboard
Copy ...	Copies the selection to the clipboard
Paste ...	Inserts the content of the clipboard at the selected location
Undo selection ...	Reverses the last action. It is possible to recall the command log and undo the last actions using the pull down menu
Redo ...	Repeats the last cancelled action
What's this? ...	Provides help on toolbar icons

4.2 Knowledge

Formula ... The feature *Formula* corresponds with the one of the pull down menu *Tools*

URLs and Comment ... Create and edit URL addresses.

Check Analysis Toolbox … The check analysis tool allows users to show and fix all broken checks to validate the design and generate reports

Design Table ... Create and edit design tables and laws to create and edit component families

Knowledge Inspector ... Analyzes impacts of change in parameter value or advises parameter modification

Lock Selected Parameters … Locks selected parameters and parameters in selected features

Equivalent Dimensions … Creates equivalent dimensions

4.3 Workbench

Workbench ... The *Workbench* icon indicates the active workbench

4.4 Graphic Properties

Graphical adjustments such as fill colour, zooming, line thickness, line style, point style and layer setting can be done.

4.5 View

Fly ...	When navigating in the *Fly* mode, translations and rotations in all three directions in space are possible.
Fit all in ...	Zooms in or out, so that all the selected geometry optimally fits the available space.
Pan ...	Pans the view
Rotate ...	Rotates the view
Zoom In...	Zooms in in increments
Zoom Out ...	Zooms out in increments
Normal View ...	Displays the part with a view normal to a plane
Create Multi-View …	Creates four different views in the current window
Views ...	Different standard views can be chosen: *Isometric View, Front View, Back View, Left View, Right View, Top View, Bottom View, Named Views*

View Modes:

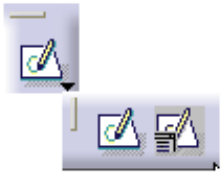
Shading …	Displays the geometry in shading mode
Shading with Edges …	Displays the shaded geometry with edges
Shading with Edges without Smooth Edges …	Displays the shaded geometry with edges without smooth edges
Shading with Edges and Hidden Edges …	Displays the geometry with edges and hidden edges
Shading with Material …	Displays the shaded geometry with material
Wireframe …	Displays the geometry in wireframe mode
Customize view parameters …	Activates the customized view mode, enabling a customization of the view parameters

Hide / Show ...	Alternatively displays hidden and shown objects. Hidden elements are dimmed grey in the specification tree.
Swap visible space ...	Makes hidden space visible again

4.6 Select

The *Select* menu offers several selection tools. Apart from a single selection, different trap selections can be chosen.

4.7 Sketcher



The sketch mode is used to create 2D contours as a basis for the following 3D modeling. A parameterization of the sketches is not mandatory. A detailed description of the sketcher follows in chapter 5.

4.8 Sketch-Based Features, Sketch-Based Features (compact)

Sketch-Based Features and *Sketch-Based Features (compact)* are required to generate 3D solid geometries.

Pad ... Creates a prism from an open or closed profile. The profile can be generated in a sketch.

Pocket ... The command *Pocket* creates a prism from a profile that is removed from a body.

Shaft / Groove ... *Shaft* creates a rotating solid from a profile and an axis of revolution. A *Groove* is a *shaft* that is being removed from an existing geometry

Hole ... Creates a hole within an existing body. The hole can also be threaded or countersunk

Rib / Slot ... Creates a rib or a slot (i.e. a removed rib) by sweeping a profile along a center curve

Stiffener ... Creates a stiffener

Multi-sections Solid / Removed Multi-sections Solid ... Creates a solid (or a removed solid) defined by several profiles and corresponding guiding curves

4.9 Dress-Up Features

The *Dress-Up Features* enable changes on existing bodies.

Edge Fillet ... Generates an edge fillet. Additionally, several other modes are available: *Variable Radius Fillet, Face-Face Fillet* and *Tritangent Fillet*

Chamfer ... Creates a Chamfer by removing or adding material from a selected edge. Several input modes are possible (*Length – Angle, Length – Length*)

Draft Angle ... The commands *Draft Angle, Reflection Line* and *Variable Angle Draft* facilitate the creation of drafts on existing solids.

Shell ... Creates a shell by hollowing out an existing geometry

Thickness ... Selected surfaces of an existing solid can be supplied with allowances

Thread / Tap ... Creates a thread or tap by specifying its support, limits and numerical values

Remove Face … Removes one or more faces

4.10 Advanced Dress-Up Features

The command *Advanced Draft* offers enhanced draft options such as defining several pulling directions for one solid.

4.11 Reference Elements, Reference Elements (extended)

Reference elements are generated by means of prompt windows to define all relevant parameters.

Point ... Creates one or more points in space

Line ... Creates a line in space

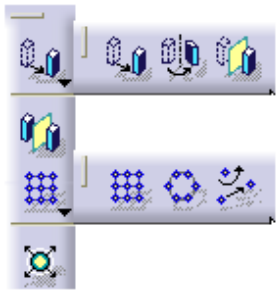
Plane ... Creates a plane in space

4.12 Constraints

Constraints Defined in Dialog Box ... Manages predefined constraints

Constraint ... Creates a constraint

4.13 Transformation Features



Translation ... Translative movement of a solid in space [*Direction, Distance*]

Rotation ... Rotates a solid around an axis [*Axis, Angle*]

Symmetry ... Mirrors a solid without duplication in reference to a selected face or plane [*Reference* = face/plane]

Mirror ... Mirrors a solid (with duplication) in reference to a selected face or plane [*Reference* = face/plane]

Rectangular Pattern ... Creates a two dimensional rectangular pattern to repeat a feature [*Instances, Spacing*]

Circular Pattern ... Creates a circular pattern to repeat a feature [*Instances, Angular Spacing*]

User Pattern ... Creates a user pattern to repeat a feature

Scaling ... Scales (expands or compresses) an element

4.14 Surface Based Features, Surface Based Features (Extended)

Split ... Splits a solid by use of a plane, face or surface

Thick Surface ... Creates a thick surface based on a surface by specifying two thicknesses

Close Surface ... This feature closes surfaces (e.g. surfaces that were designed in *Wireframe and* Surface mode), i.e. it generates a solid from the surface

Sew Surface ... Integrates surfaces into a solid

4.15 Insert

Insert is used to insert a new body or geometrical set in the specification tree. The new element is inserted beneath the active element or into a specified component.

4.16 Boolean Operations

By means of this menu commands affecting two bodies can be carried out. The reference body should be set *In Work*.

Assemble ...	Assembles a body with another body
Add ...	Adds a body to another body
Remove ...	Removes a body from another body
Intersect ...	Intersects a body with another body, resulting in a single body that displays the shared space

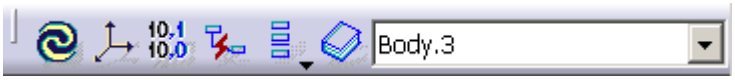
Union Trim ... Merges two bodies and enables a trim function

Remove lump ... Removes a single piece of a body. This is a special case of *Boolean Operations* as it concerns only one body

4.17 Selection Sets

Selection Sets Edition ...	Create and edit selection sets
Selection Sets ...	Management of the saved selection sets
Find Owning Selection Sets …	Find all selection sets including the selected element

4.18 Tools



Update All ...	Updates all features and connections within the part
Axis System ...	Creates an axis system
Mean Dimensions ...	Computes mean dimensions on toleranced parameters
Create Datum ...	Creates a datum feature (= feature without history)
Only Current Body …	Option to display only the current body
Catalog Browser ...	Opens a catalog, e.g. a screw catalog
Select Current Tool …	Selects / renames a current tool

4.19 Annotations

Text with Leader ...	Creates a text with a leader line
Flag Note with Leader ...	Creates a flag note with a leader line and URL support.

4.20 Analysis

The *Analysis* features support a construction check regarding the producibility.

Draft Analysis ...	Analysis of drafts
Curvature Analysis ...	Analyzes the curvature of surfaces
Tap - Thread Analysis ...	Analyzes all threads and taps of a component

4.21 Apply Material

Material properties can be applied to a body, enabling the computation of weight, inertia etc..

Apply Material ... Applies a material to a part

4.22 Measure

Measure Between ...	Measures between two elements
Measure Item ...	Measures characteristics of an element
Measure Inertia ...	Measures inertial properties associated to a selected volume

5 The sketch mode *Sketcher*

The sketch mode is used to create two dimensional sketches. A parameterization is not mandatory. When working in the workbench *Part Design*, sketches can serve as a basis for the generation and modification of solids.

5.1 Using the Sketcher

The sketch mode is activated by clicking on the button *Sketch*. The *Sketch Support* has to be a plane or a planar surface. The *Sketcher* rotates the selected plane parallel to the screen plane (default setting in the *Options*).

For switching or refreshing the adjustment of the screen view, the feature *Normal View* has to be used. The image plane is aligned parallel to the selected support plane.

In sketch mode a reference coordinate system is laid into the chosen plane. The sketch module is positioned just below the active object in the *Specification Tree*, and it contains the *Geometry* and the *Constraints.*
A grid is shown, offering a snap function, if *Snap to Point* has been activated. The preset toolbars are displayed on the right margin.

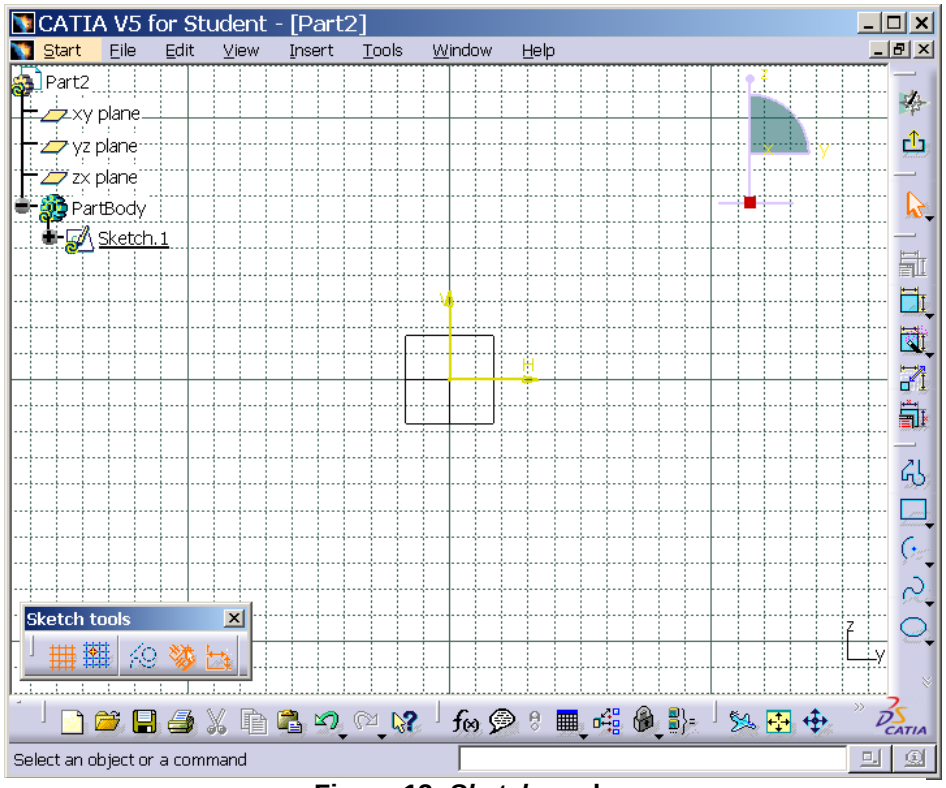


Figure 13: *Sketch* mode

The individual setup of the desktop is done through *Tools / Options* (in the menu bar). The selection of the toolbars happens via the menu *View / Toolbars.*

5.2 Operations in the sketch mode

The sketch mode contains, apart from standard toolbars, the following workbench-specific tools:

5.2.1 Sketcher

Workbench – *Sketcher* icon... Shows the active workbench

Exit workbench ... Leaves the *Sketcher* and gets back to the previously active workbench

5.2.2 Profile

The menu *Profile* provides features for the creation of basic geometrical elements. While not being parameterized, the contour is displayed as white lines.

Profile ... Creates a profile made of lines and arcs.

Predefined Profile ... Creates predefined profiles:

-*Rectangle*
-*Orientated Rectangle*
-*Parallelogram*
-*Elongated Hole*
-*Cylindrical Elongated Hole*
-*Keyhole Profile*
-*Hexagon*
-*Centered Rectangle*
-*Centered Parallelogram*

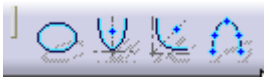
Circle ... Creates circles and parts of circles:

-*Circle*
-*Three Point Circle*
-*Circle Using Coordinates*
-*Tri-Tangent Circle*
-*Three Point Arc*
-*Three Point Arc Starting with Limits*
-*Arc*

Spline ... Creates a spline by clicking or selecting points:

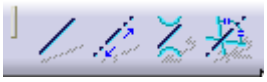
-*Spline* (curve through points)
-*Connect* (Creates an arc connecting two curves)

Conic ... Creates Conic Curves:



-Ellipse
-Parabola by Focus
-Hyperbola by Focus
-Conic

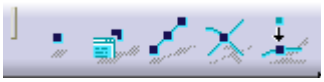
Line ... Creates Lines:



-Line
-Infinite Line
-Bi-Tangent Line
-Bisecting Line
-Line Normal To Curve

Axis ... Creates an axis, e.g. for the creation of rotating bodies

Point ... Creates a point by clicking:



-Point by Clicking
-Point by Using Coordinates
-Equidistant Points
-Intersection Point
-Projection Point

5.2.3 Operation

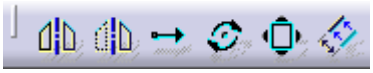
Corner ... Creates a corner with a user defined radius.

Chamfer ... Creates a beveled corner.

Relimitations ... Modifies lines or profiles

-Trim
-Break
-Quick Trim
-Close
-Complement

Transformation — Transformation components:



-Mirror
-Symmetry
-Translate
-Rotate
-Scale
-Offset

3D Geometry ... — Generates 2D-curves from 3D elements:

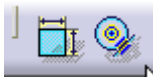
-Project 3D Elements
-Intersect 3D Elements
-Project 3D Silhouette Edges

5.2.4 Constraint

The toolbar *Constraint* contains features for the assignment of constraints.

Constraints Defined in Dialog Box ... Creates constraints checked in a dialog box

Constraint ... — Creates a geometrical or dimensional constraint



-Constraint
-Contact Constraint

Constrained Geometry ... — Creates Constraints:

-Fix together
-Auto Constraint

Animate Constraint ... — Animates dimensional constraints to show how the constrained system reacts

Edit Multi-Constraint — Edits constraint values and evaluates the constrained geometries at the end

5.2.5 Sketch Tools

Grid ... — Displays a grid

Snap to Point	Snaps the points to the nearest intersection points of the grid
Construction / Standard Element ...	Converts sketch elements into ‚construction' or ‚standard' elements
Geometrical Constraints ...	Creates the detected and the internal constraints during sketching
Dimensional Constraints ...	Creates dimensional constraints

5.2.6 Tools

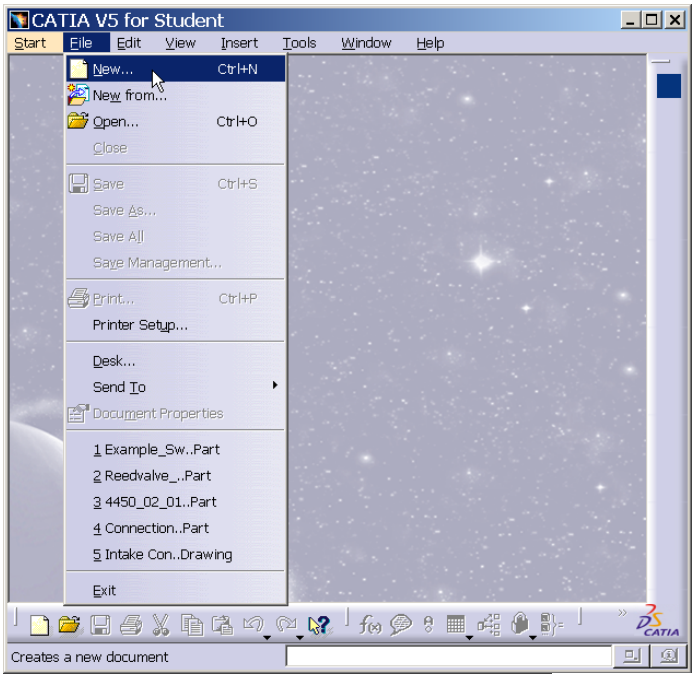
Create Datum …	Creates a datum feature (without history)
Only Current Body …	Option to display only the current body
Output Feature …	Creates an output feature by selecting a 2D geometry
Profile Feature …	Creates a profile feature by selecting a 2D geometry
2D Analysis Tools …	Tools assisting the sketch analysis: *-Sketch Solving Status* *-Sketch Analysis*

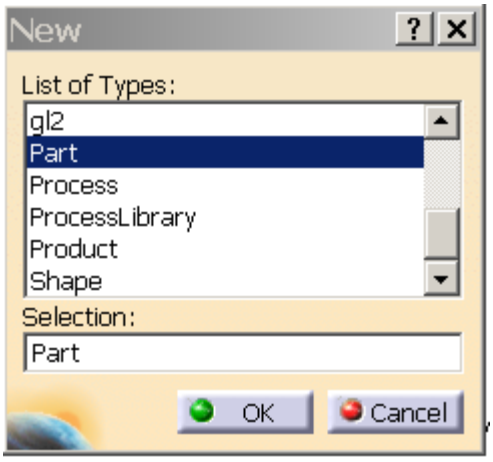
Example 1: Regular hexagon, wrench size of 100mm

Intention: Using the Sketcher

This first example should describe how to generate sketches within the *Part Design* workbench.
To activate *Part Design* just open a new part after having started CATIA V5:

1. File / New





By selecting *Part* the workbench *Part Design* opens. The predefined toolbars of the selected workbench appear around the working area on the screen.

Figure 14: Opening a new part

2. Open the *Sketcher*

3. Select the xy-plane as sketch plane

The sketch plane can be selected by clicking on a plane either in the modelling area or in the structure tree.
The selected plane is then rotated in a way that it is parallel to the screen plane, and the sketch mode is activated. The corresponding toolbars appear.

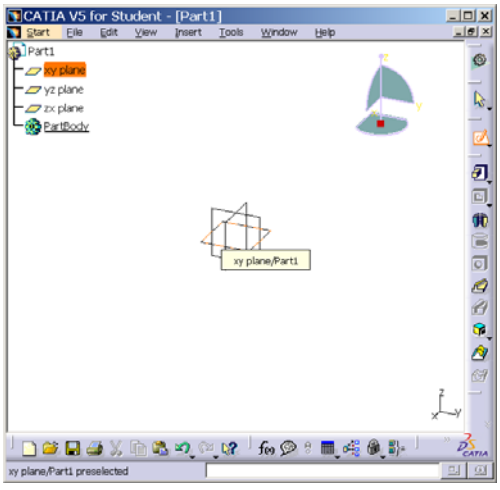


Figure 15: Select a sketch plane

4. Create a sketch

Using the feature *Line* (contained in the *Profile* menu), a hexagon can be drawn. Double clicking the icon activates the repetition mode. The *Snap to Point* mode allows to catch the ending point of the previous lines. When the line happens to be nearly vertical or horizontal, a corresponding constraint is established by activating the *Snap to Point* mode. The repetition mode of the *Line* feature is deactivated by clicking on the icon once more. Another way of creating curves containing lines (and circles) is provided by the feature *Profile*.

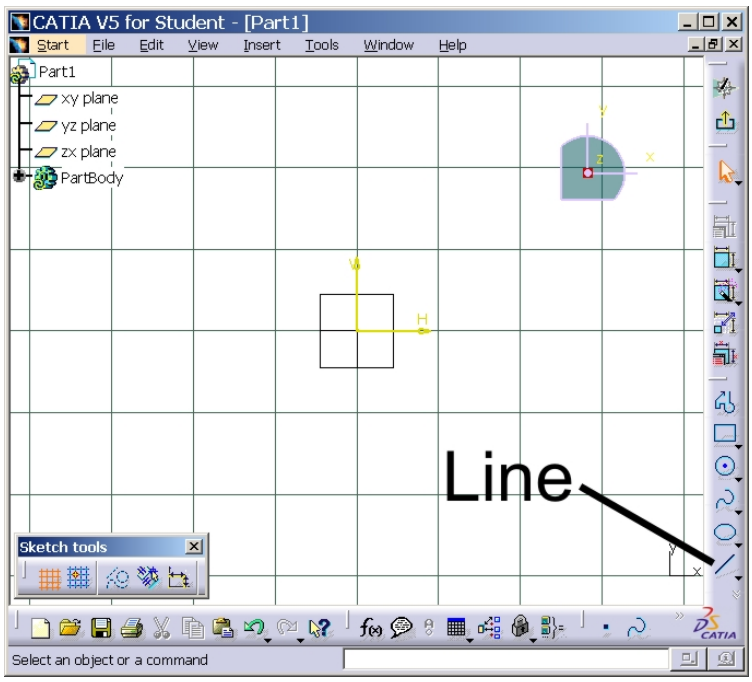


Figure 16:The *Sketch* workbench

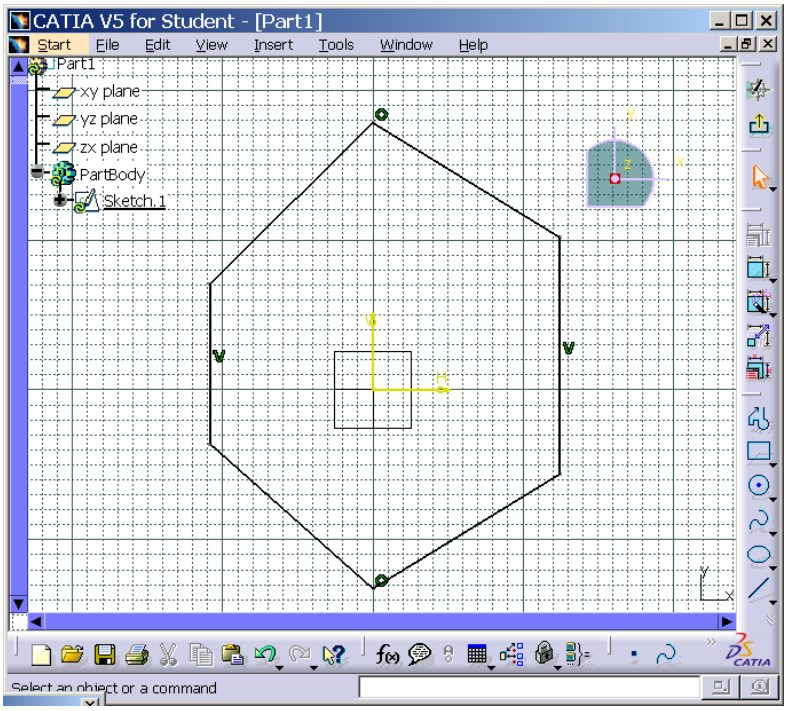


Figure 17: The first draft of the hexagon

The symbols *H* and *V* next to the lines designate their horizontal or vertical orientation. During sketching the lines turn blue to show a constraint. Coincidences are displayed as small green circles. By double clicking onto the constraints, the corresponding windows open. Constraints can be removed using the delete function.

The geometrical elements can be defined in different ways. One possibility is by double clicking the geometrical elements opening an input window.

Figure 18 shows the input window of a vertical line. Similar windows exist for all basic geometrical elements in the sketch mode.

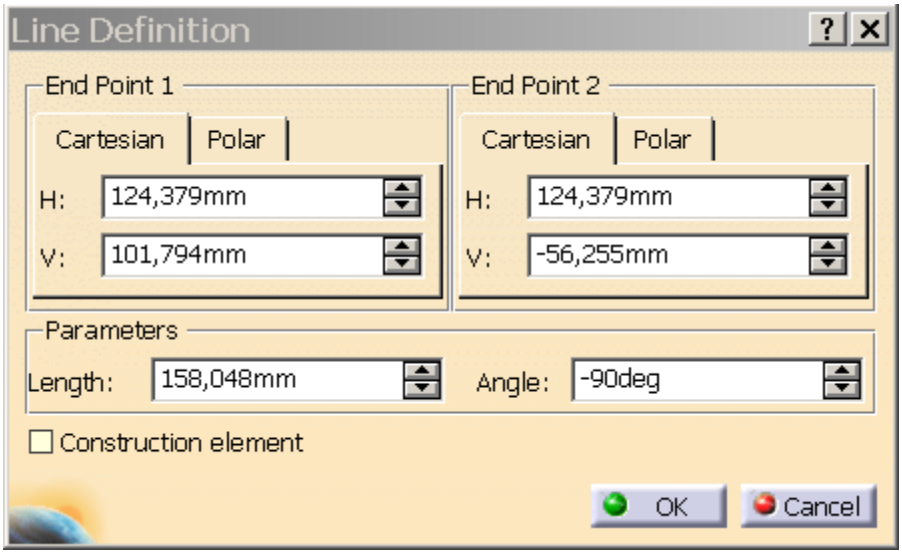


Figure 18: Definition of the vertical line on the right side

The feature *Construction Element* enables the creation of auxiliary sketches or elements that are NOT used for the generation of bodies. Auxiliary elements have to be created as *Construction Elements*; otherwise, features as *Pad* are not able to create a geometry from the sketch.

5. Constraining the sketch

Using *Constraints,* different constraints (geometrical constraints and dimensions) can be defined. Both, angle dimensions and linear dimensions, can be defined.

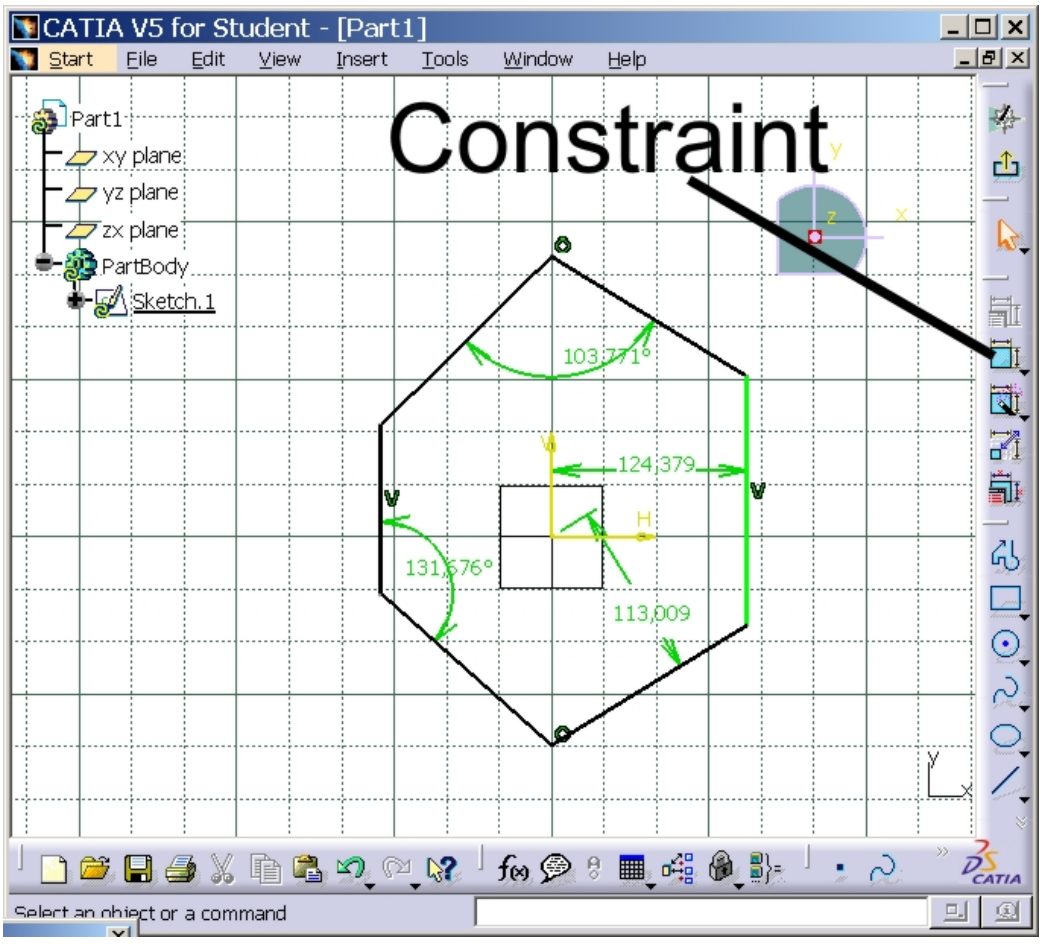


Figure 19: Using *Constraints*

One possibility to dimension the hexagon is shown in Figure 21, but several other ways are possible. The dimensions shown in Figure 19 show the values of the first draft; the dimension values can be changed by double clicking the values, e.g.:

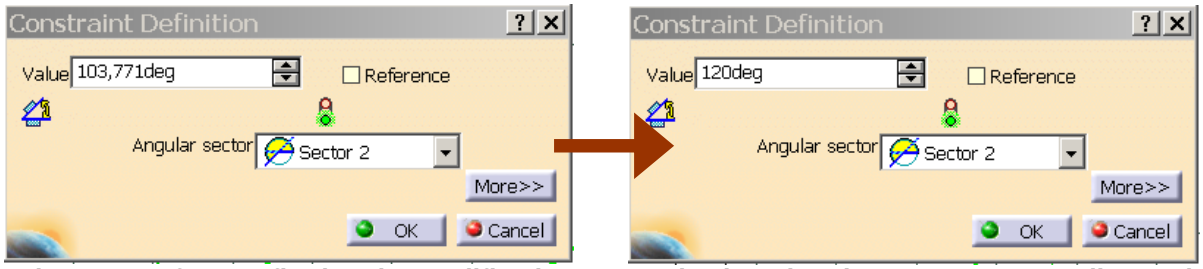


Figure 20: After confirming the modification, CATIA is changing the geometry according to the new value.

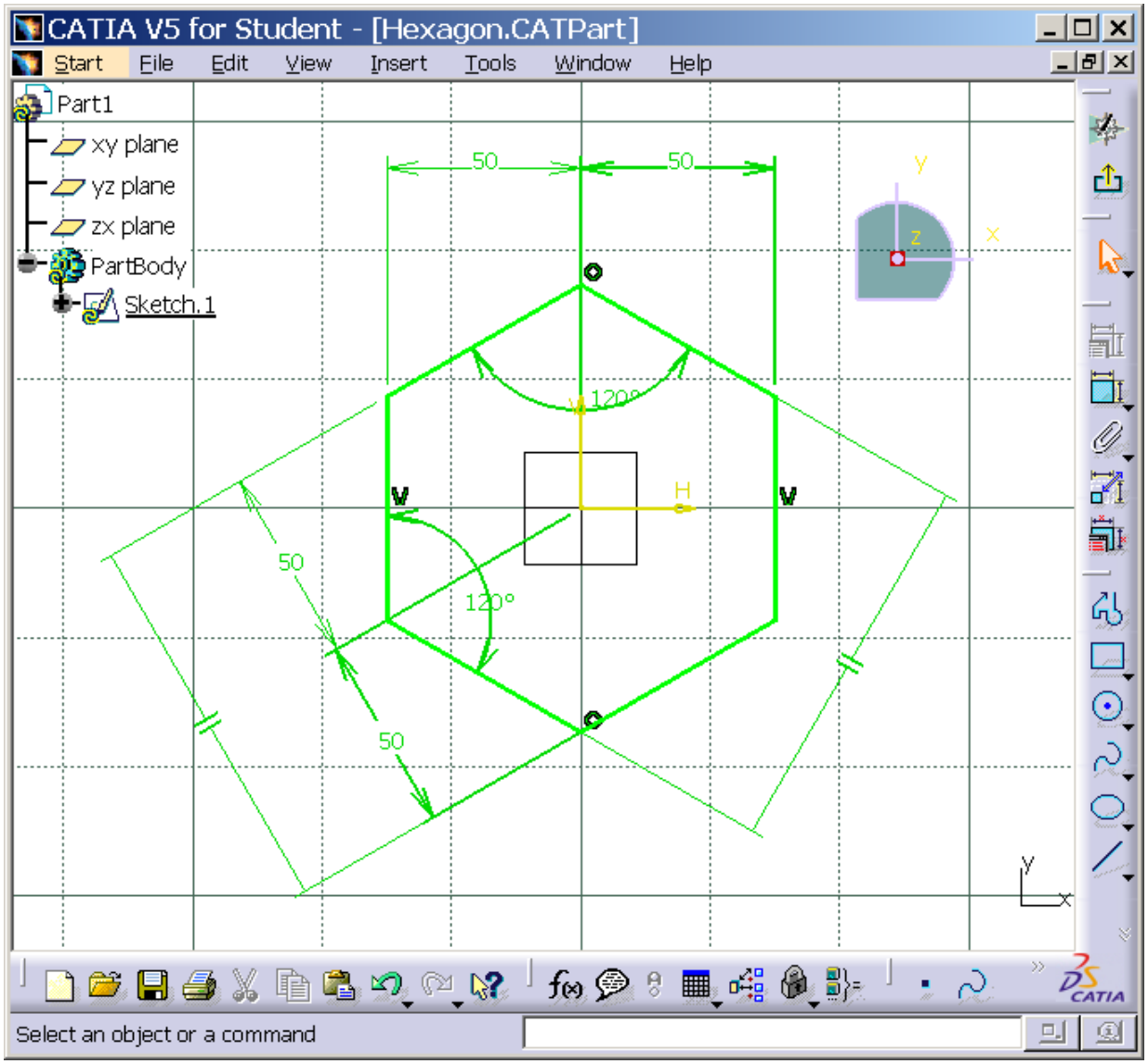


Figure 21: The completely dimensioned hexagon

The hexagon has to be constrained as well as its position in the working space. The dimensioning of the sketch is complete when all elements turn green. White lines are not completely parameterized, purple lines are overconstrained.

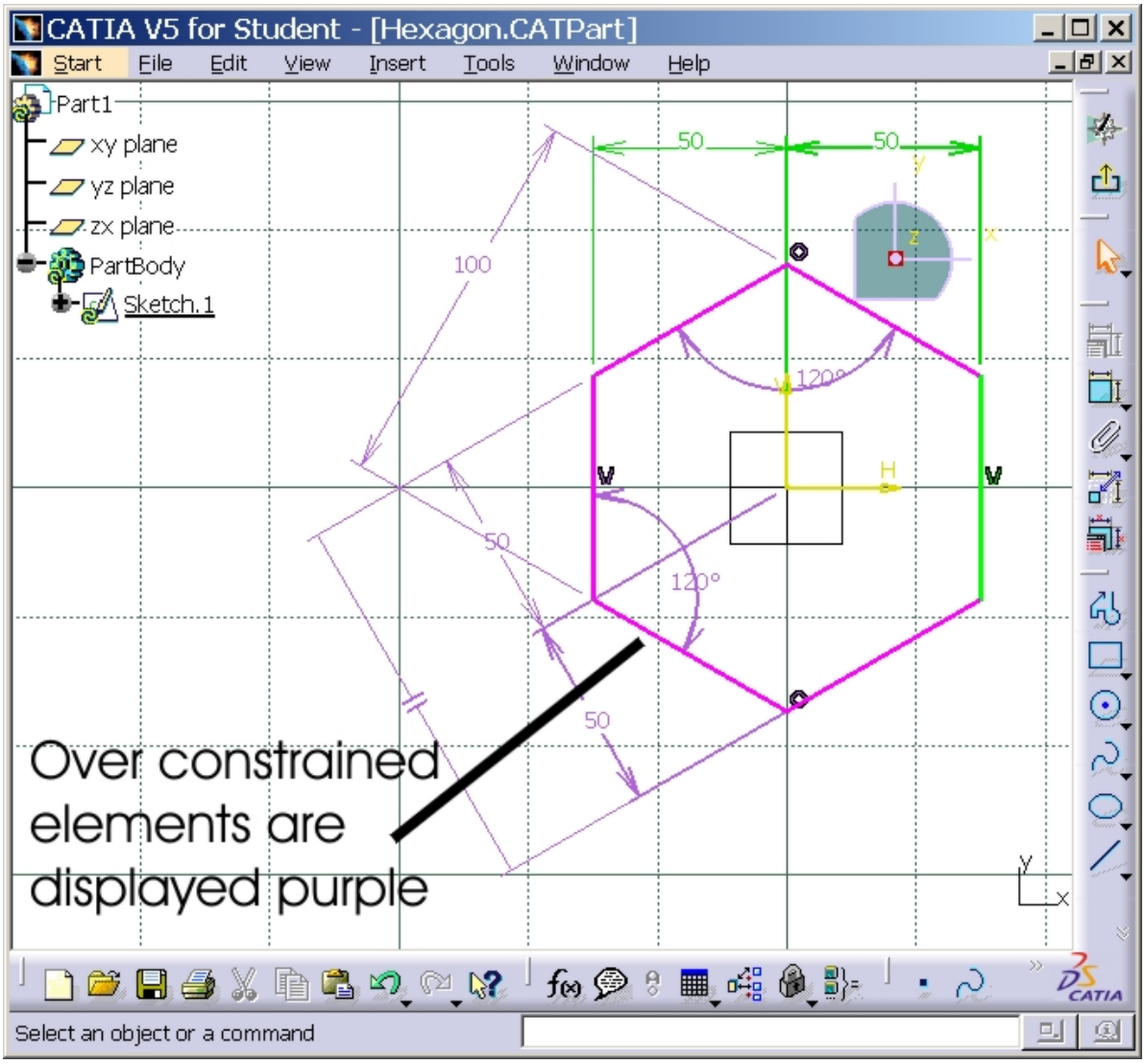


Figure 22: Sketch containing over constrained elements

Overconstrained elements have to be revised. Underconstrained elements can be used, but undefined dimensions are considered as variable. Therefore these elements (displayed white) can be modified by CATIA, if needed.

In case that the profile is opened, it can be closed using *Trim*. After selecting the button the two geometrical elements that shall be trimmed have to be selected.

The sketch mode offers the following constraints:

Horizontal
Vertical
Coincidence
Perpendicular (90°)
Fix
Parallelism

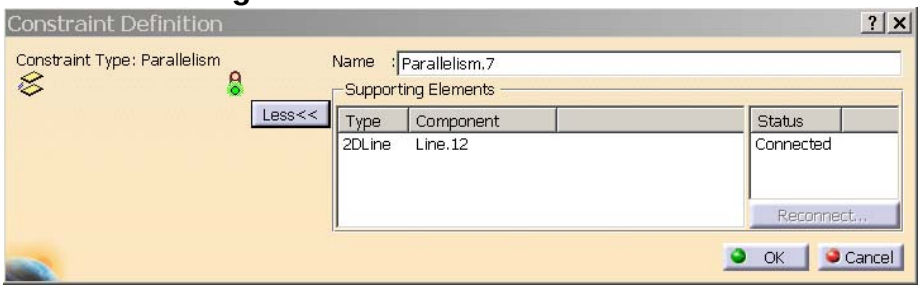


Figure 23: Constraint definition

Concentricity
Tangency
Length, Distance, Angle, Diameter, Radius

The pop up window shows all necessary inputs for the particular constraint. This feature can be useful especially for the modification of dimensions. Some definition windows are shown in Figure 24. They are all similar: The specifications of the geometrical elements are displayed and can be modified.

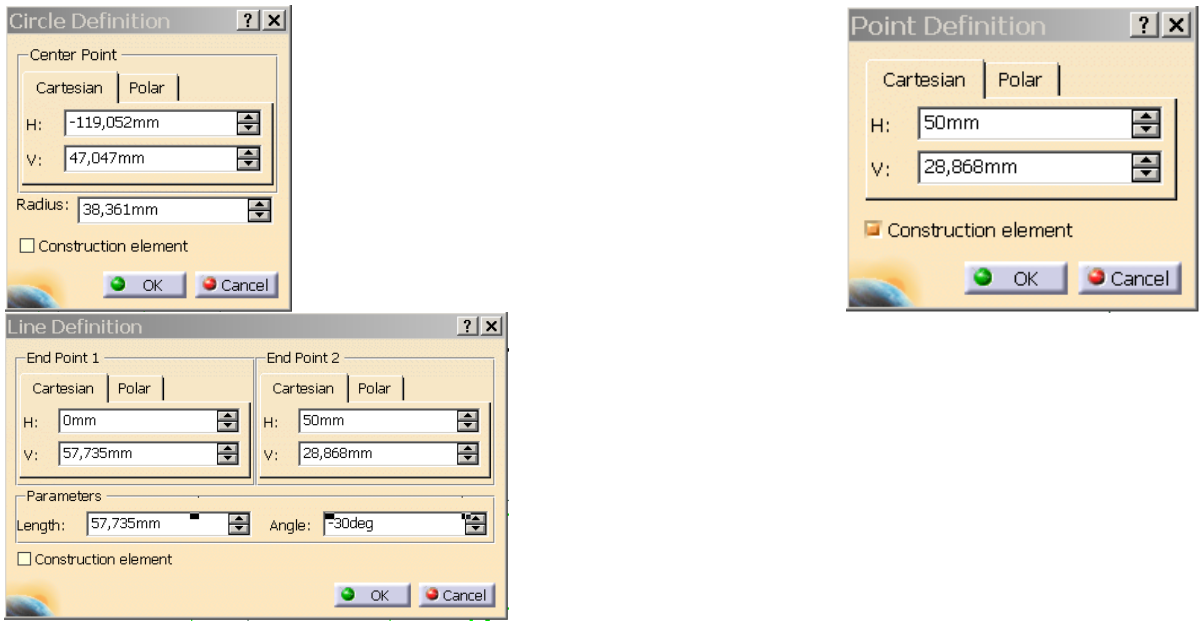


Figure 24: Definition windows for several geometrical elements

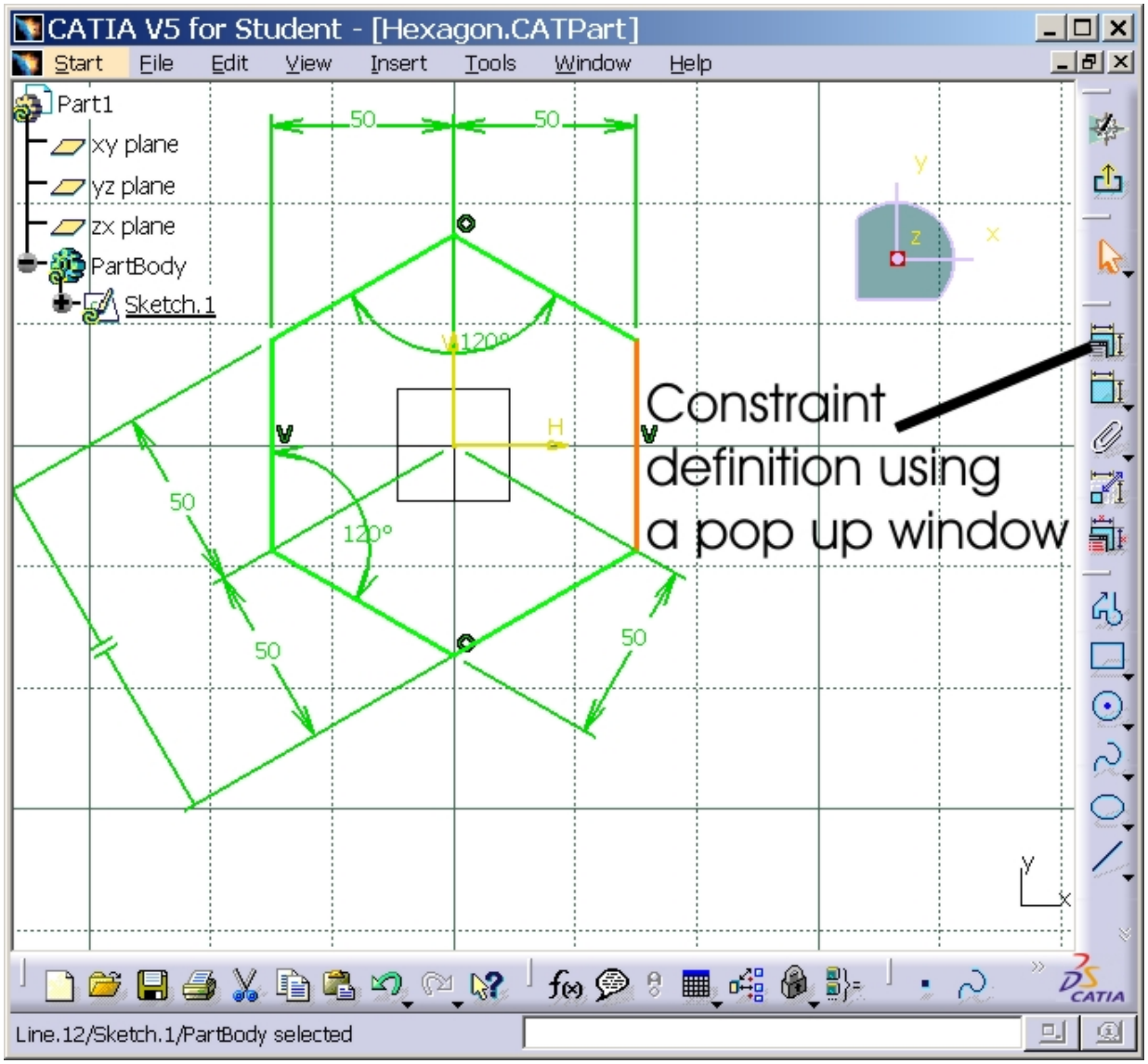
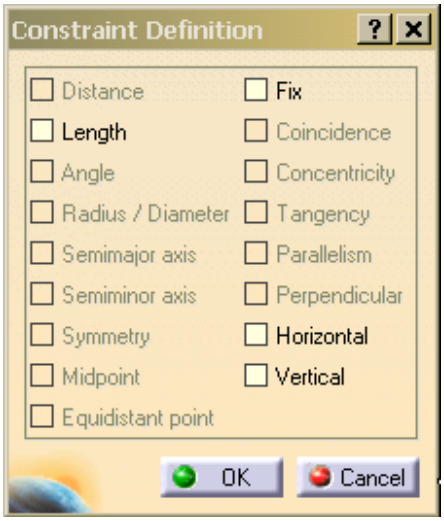


Figure 25: Constraint definition via a definition window

Apart from generating dimensions one by one, it is possible to create several constraints at one time. To open the according window, the regarding elements have to be marked (left mouse button, for multi selection press the Ctrl. button on the keyboard) and *Constraints Defined in Dialog Box* selected.



The dialog box allows to select constraints that can be used for the selected elements.

The button *Exit Workbench* causes CATIA to return into the 3D mode.

Alternative hexagon design:

CATIA provides the feature *Hexagon,* which creates a regular hexagon.

The required input data are the center point of the hexagon and the center point of one of the edges. The wrench size and the orientation of the hexagon can be defined by use of constraints. This special feature creates not only the hexagon contour, but all the single elements that can be selected and modified individually. For example, this permits to delete one single edge.

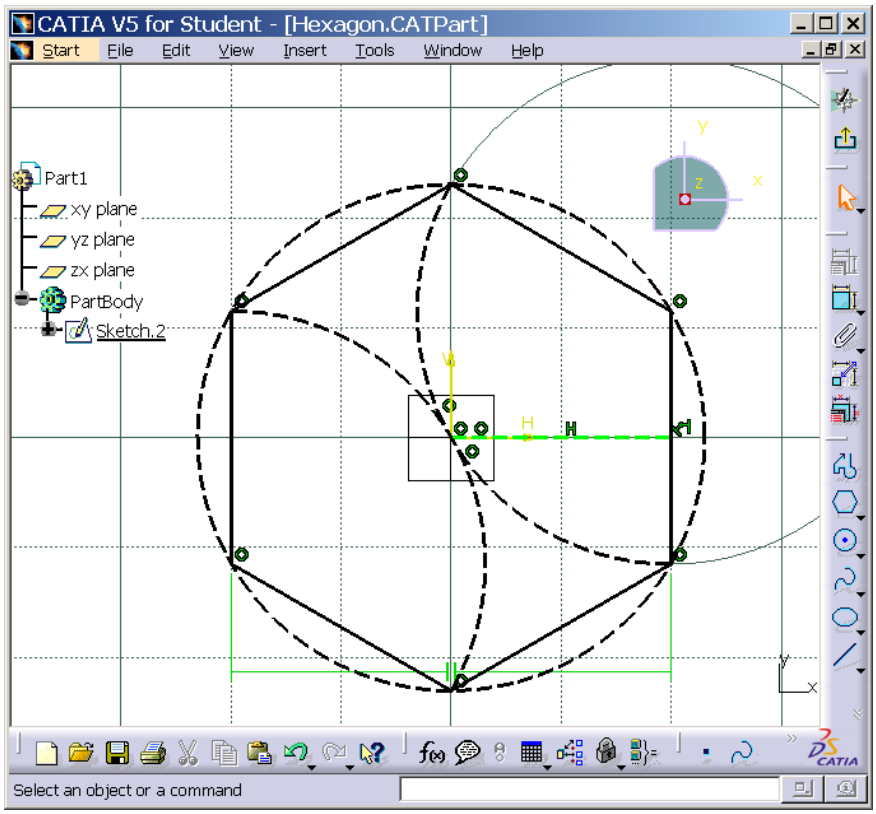


Figure 26: Create a hexagon using *Hexagon*

The toolbar *Sketch tools*

Sketch tools is used to set miscellaneous helpful adjustments. *Grid* displays gridlines; the grid size can be adjusted in the menu *Tools* / *Options* / *Mechanical Design* / *Sketcher* / *Grid*. *Snap to Point* snaps points to the nearest intersection points of the grid.

The next feature switches to the construction mode. When *Construction* / *Standard Element* is activated, from now the generated geometry is being defined as construction elements and can not be used in the sketch mode. Elements can also be defined as construction elements ex post by simply selecting them and pressing the *Construction* / *Standard Element* button. Another possibility to switch between the two modes is to open the definition box of the respective element and activate / deactivate *Construction Element*.

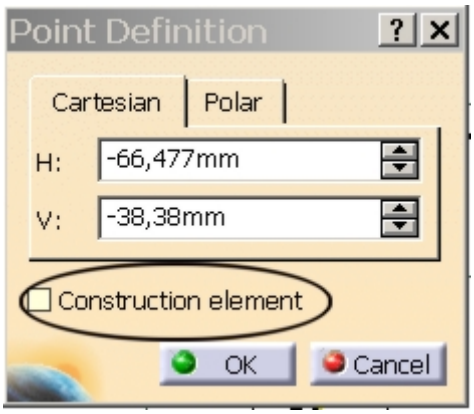


Figure 27: Turning a point to a *Construction Element*

The following features activate the automatic assignment of geometrical or dimensional constraints. These constraints are displayed green and support the sketching. If *Geometrical Elements* is being deactivated, the constraints are displayed but not applied.

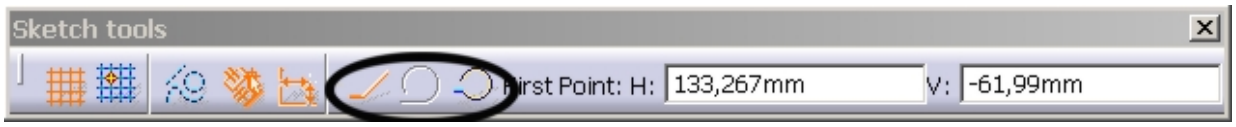


Figure 28: *Sketch Tools* toolbar for the feature *Profile*

Some additional options supporting the creation of geometry are contained in *Sketch tools*. One of them is activated, if the feature *Profile* is used: The type of the profile continuation can be chosen (*Line, Tangent Arc, Three Point Arc*).

Sketch Analysis

Sketch Analysis enables an easy check of the sketch concerning open contours, interfering points or overlaps. This tool can be used for revising and editing the sketch.

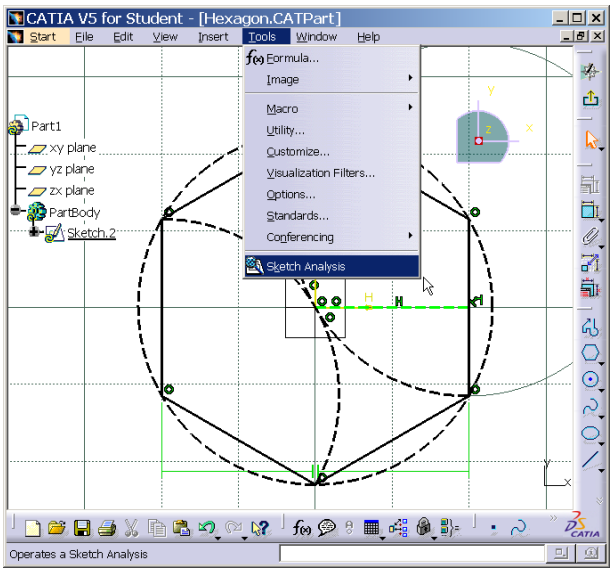


Figure 29: *Sketch Analysis*

5.3 Structure of the specification tree of a sketch

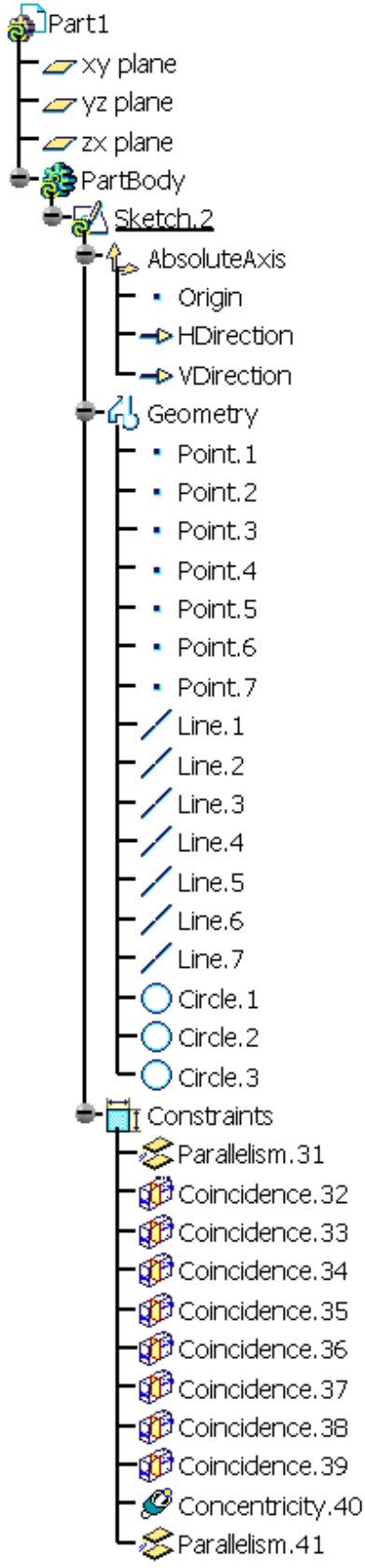


Figure 31: Specification tree of a sketch

The *Specification Tree* shows the position of the sketch within the *Body* and all the elements contained in the sketch. The coordinate system of the sketch is displayed in the tree as well. When leaving the sketch mode, the coordinate system is being switched into the noshown space.

The group *Geometry* contains the geometrical elements; double clicking the elements in the tree opens the definition windows of the according elements.

Constraints contains the set constraints; they can be edited similar to the geometrical elements (by double clicking).

Clicking on an element with the right mouse button opens a window with multiple options (Figure 30).

Figure 30: Option window of a line

Apart from editing features such as *Hide / Show* and *Cut / Copy / Paste*, the menu offers access to *Properties* (*Graphic, Feature Properties* and *Mechanical*) where, amongst other things, the name and the graphic properties of the element can be set.

Figure 32 shows the complete hexagon sketch. Using *Pad* generates a body from the sketch.

For editing the sketch, e. g. modifying the geometry, the sketch has to be double clicked, either in the structure tree or in the modeling area. The changes of the sketch are automatically taken into account for the creation of the pad.

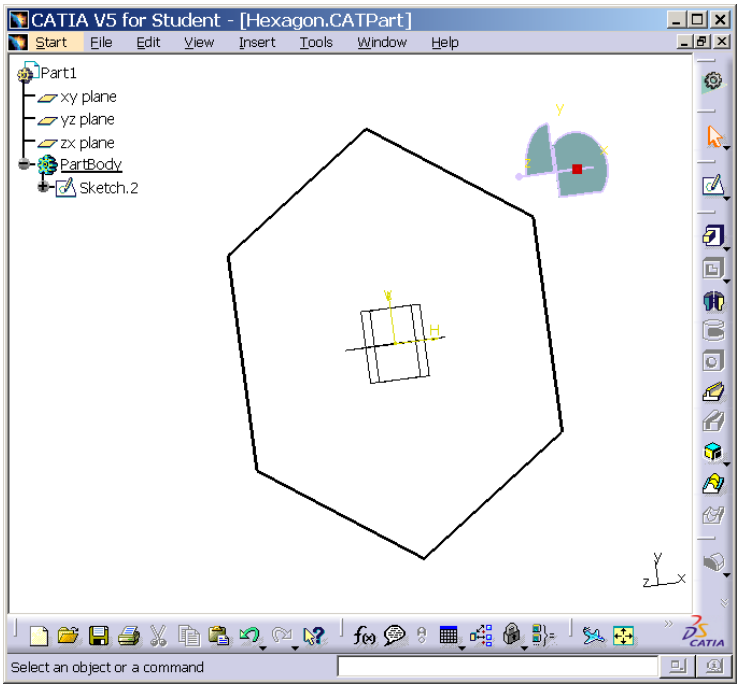


Figure 32: Complete hexagon

Example 2: Milled panel

Intention: Contour creation and dimensioning in the Sketcher

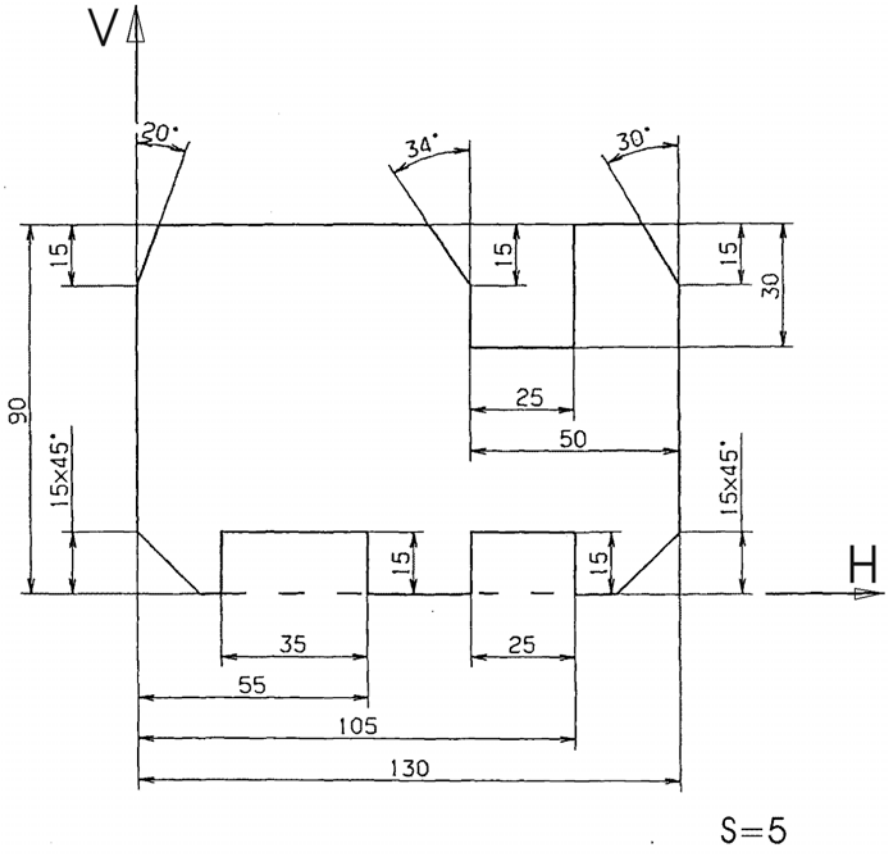


Figure 33: Milled panel

Example 3: Mounting plate

Intention: Contur creation and dimensioning in the Sketcher

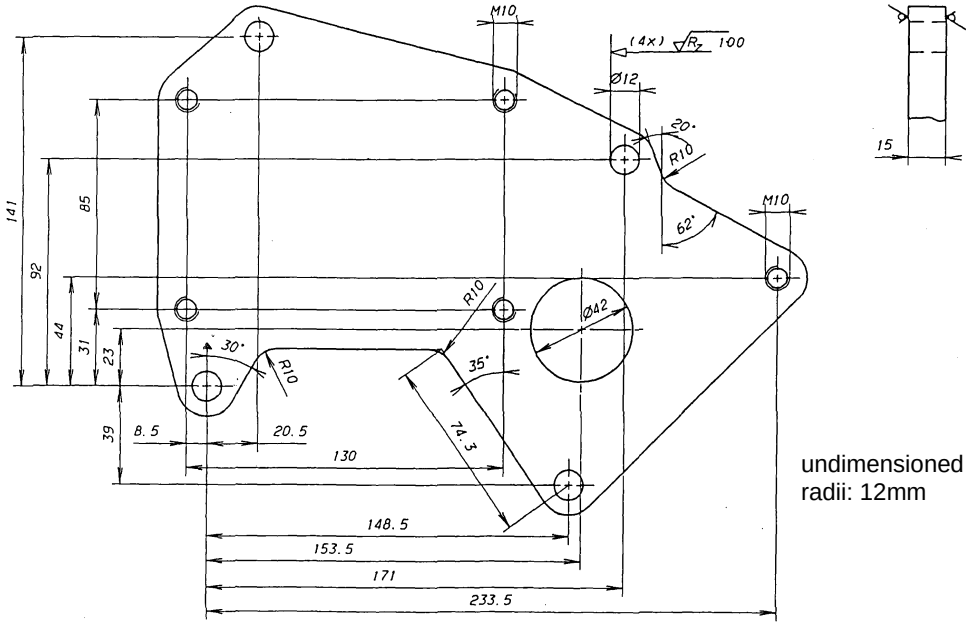


Figure 34: Mounting plate

6 Generation of bodies in the workbench *Part Design*

The workbench *Part Design* is used to create solid bodies. One *Part* can contain several bodies. Based on the bodies, other features can be carried out, such as *Drafting* or the creation of *Products*. The bodies are generated by use of sketches (created in the *Sketcher* mode). Based on these contours, basic solids are designed. Subsequently the solids can be modified (e. g. with *Pocket, Chamfer, Fillet, Hole etc.).*

6.1 3D Basic Features

The following basic features are offered:

Pad

Shaft

Rib

Example 4 - PAD: Hexagon profile, Wrench size 100mm, Height 20mm

Intention: Application of the feature Pad

The sketch created in Example 1 is used as a basis for the solid. Selecting the button *Pad* opens a definition box wherein the attributes can be edited.

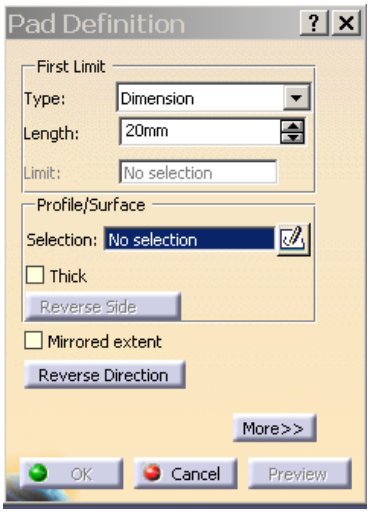


Figure 35: *Pad* definition

The *Limit Type* specifies the definition of the pad length. One possibility is to use limiting planes or surfaces. This example uses the limiting type *Dimension*, the length is set 20mm. *Sketch 1* is selected as a profile.

☺ **Annotation:** The sketch has to contain a closed contour to create a standard pad. The feature *Mirrored extent* enables the extension of the body in both directions, *Reverse Direction* switches the extension direction.

Selecting *More* activates an extended definition window with the following options:

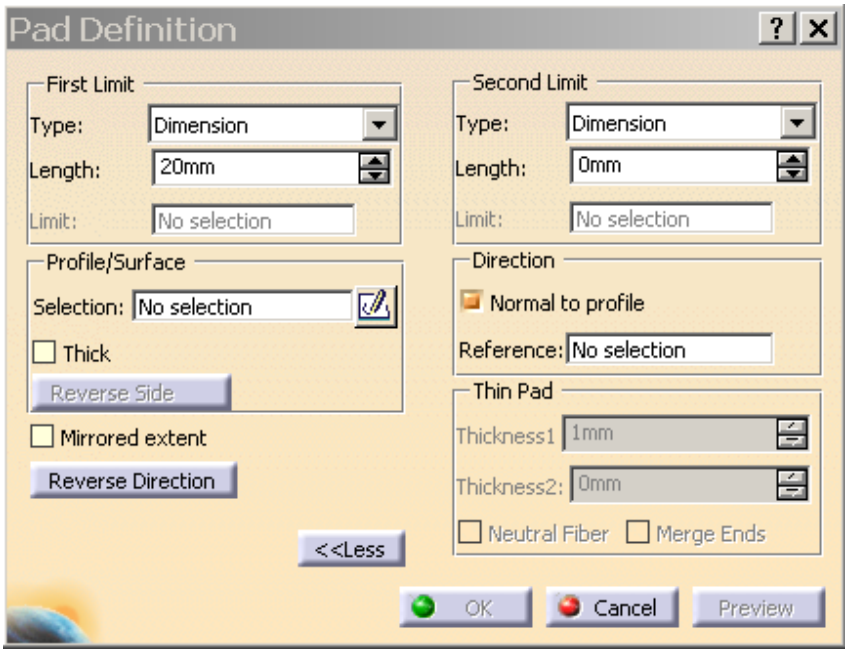


Figure 36: Extended *Pad* definiton box

Second Limit:
Extension into the other direction

Direction:
Select the extension direction (e. g. by selecting a line)

Thin Pad (only available when *Thick* is activated):
Creates a body with a defined thickness on both sides of the profile.

The complete body is displayed according the input values.

The hexagonally shaped solid is based on the sketch and the values of the pad definition. To modify properties, the solid can be selected (either in the model area or *Pad 1* in the specification tree). The definition box should appear; it also offers the modification of the according sketch ( ).

The element *Pad 1* has been inserted by CATIA in the specification tree.
To rename *Pad1*, click at the right mouse button. In the appearing properties window the menu *Properties / Feature Properties* can be used to rename the element, and *Properties / Graphic* changes the color.

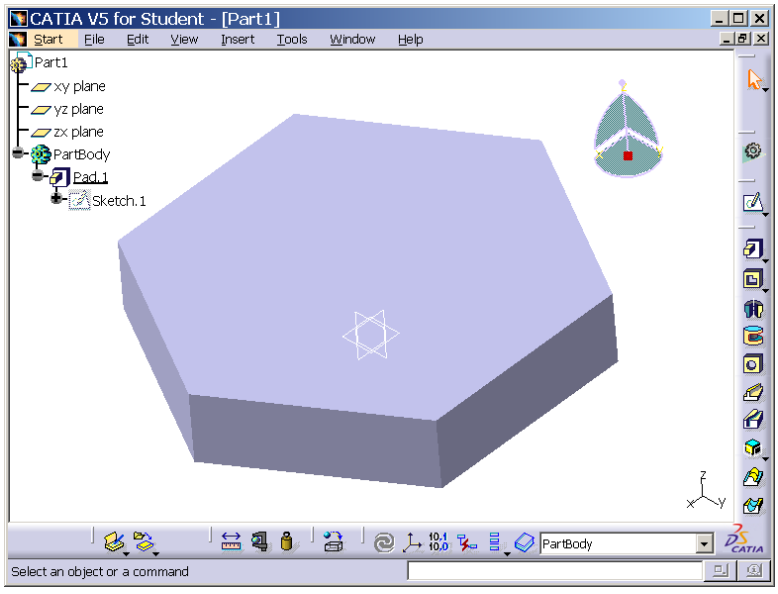


Figure 37: The complete *Pad*

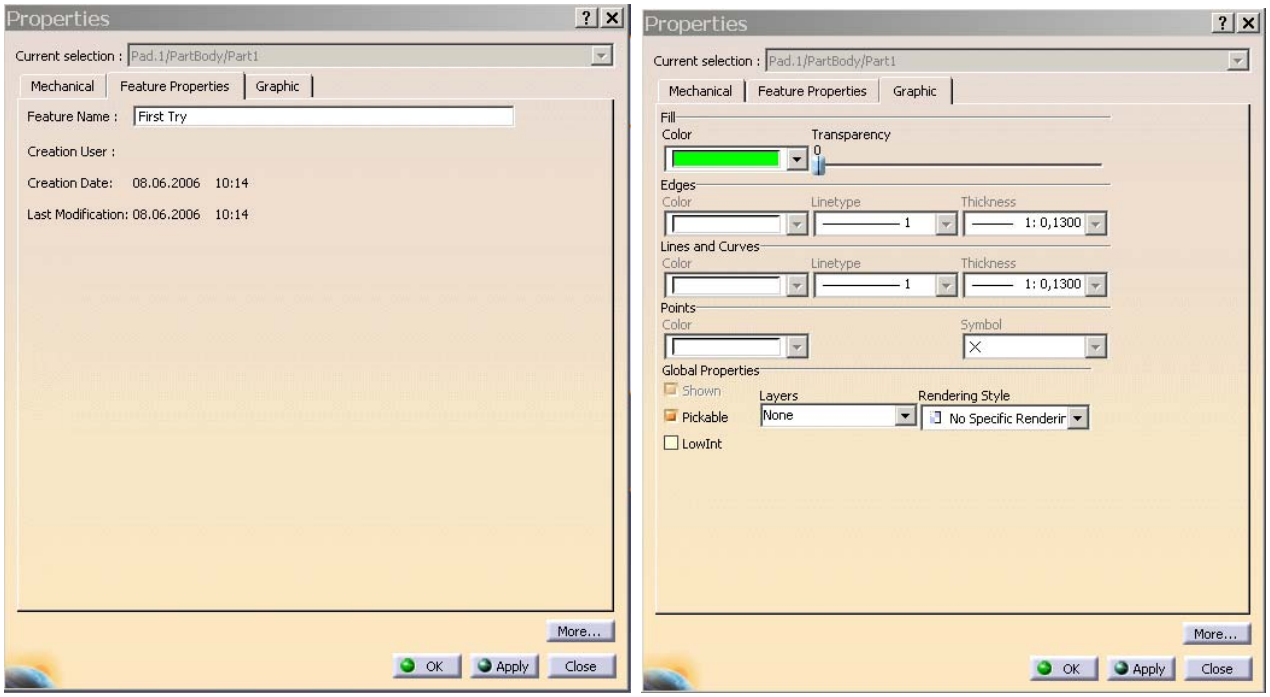


Figure 38: *Properties* menu of a pad

As a result, the hexagonal body is called *First Try* in the specification tree and is turned green.

The part can be saved using *File* / *Save as*.

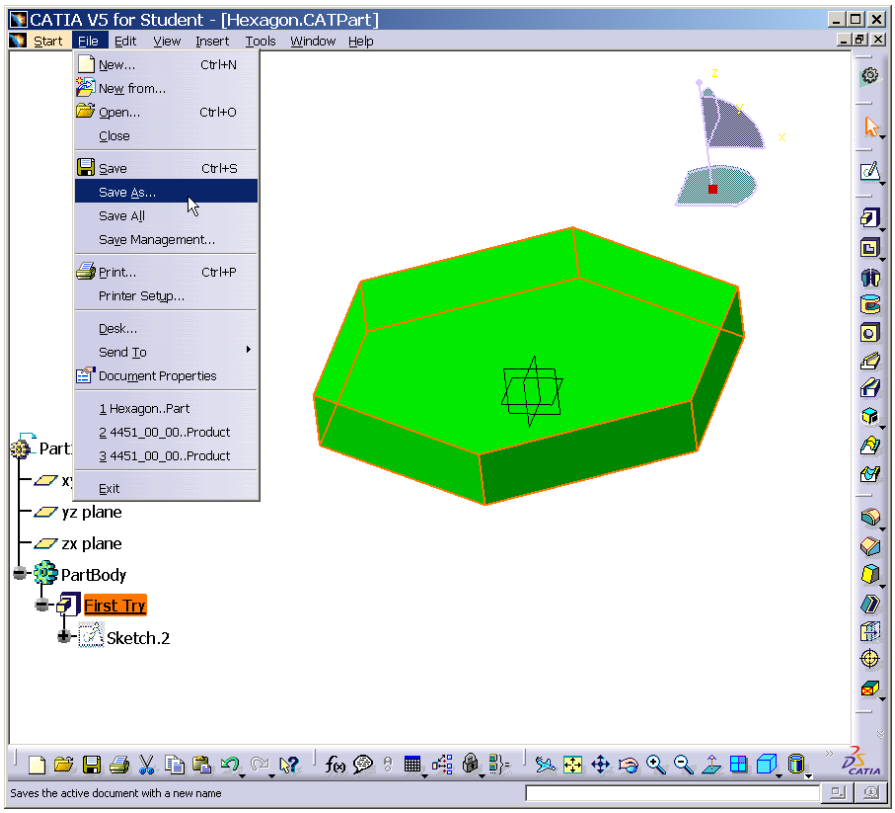


Figure 39: Saving the hexagonal part

Example 5 - SHAFT: Rotational solid

Intention: Application of the function Shaft

The feature *Shaft* generates rotating bodies; the rotation of the generating profile doesn't need to be full 360°. The rotation axis doesn't need to intersect the rotating profile, rendering possible the creation of closed rotating profiles such as tori. The *Shaft* definition demands a sketch, defining the rotating profile, and an axis of revolution.
The following profile has to be generated using the sketch mode:

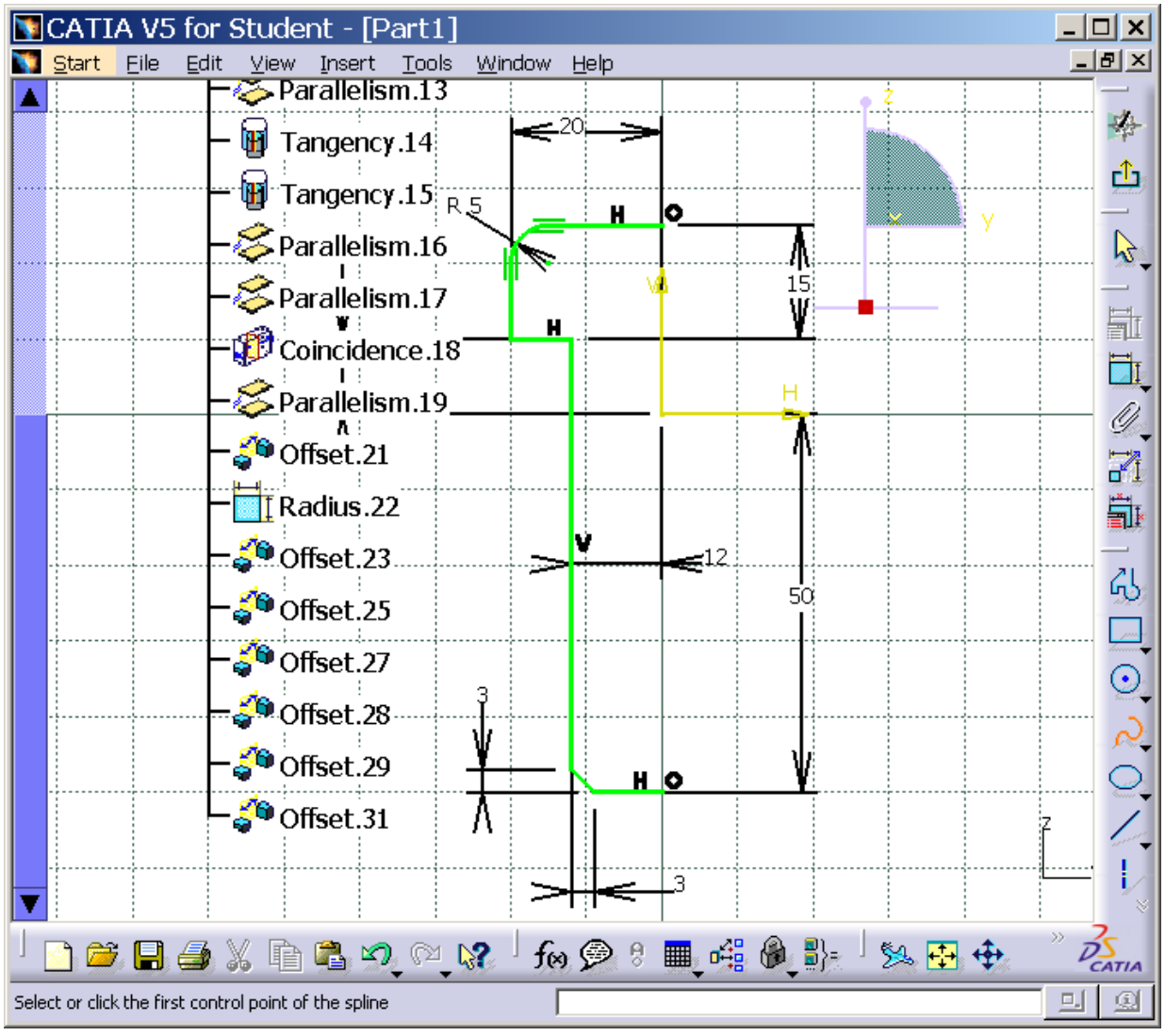


Figure 40: Sketch of the profile

After leaving the sketch mode and selecting the feature *Shaft*, a definition box for the shaft appears.

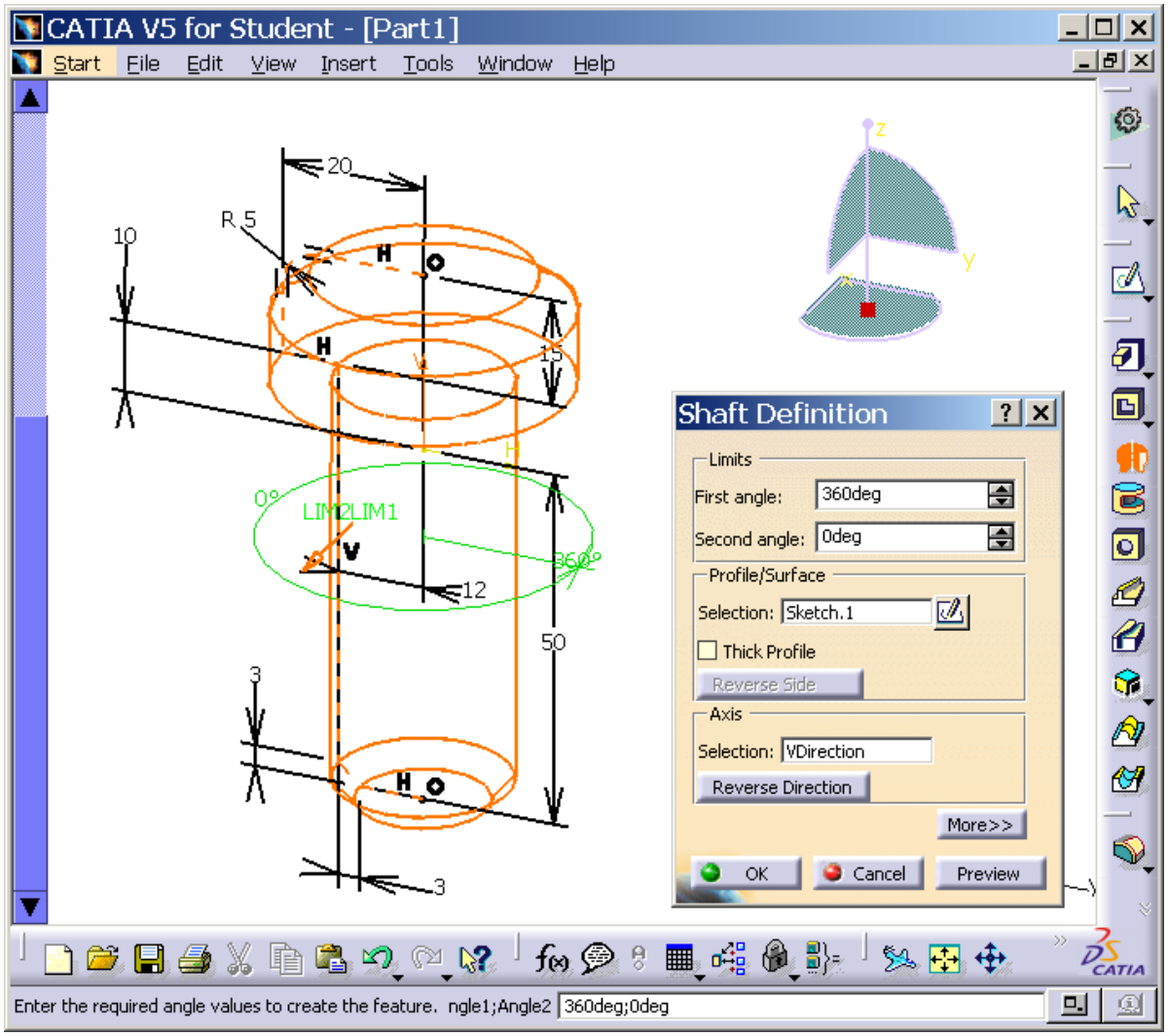


Figure 41: Creating a *Shaft*

The sketch is used as *Profile*, the rotation axis is the vertical line of the coordinate system. The *First Angle* is set to 360 degrees, the *Second Angle* is zero degree.

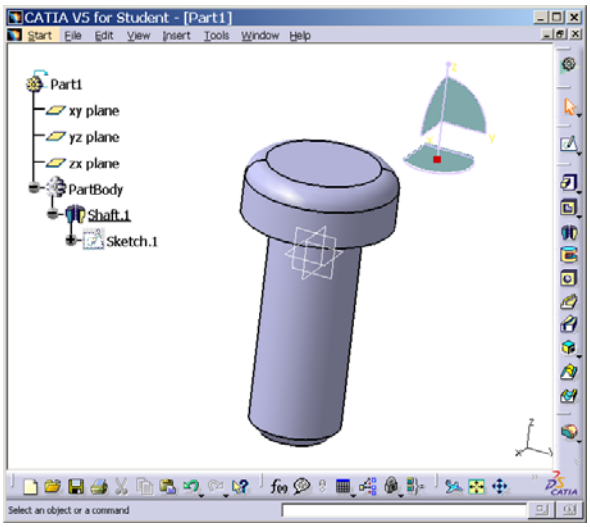


Figure 42: Complete rotating body

Example 6 - RIB: Profile swept along a center curve

Intention: Using the Rib feature

Two sketches or curves are needed to create a rib:

1. A contour of the rib
2.The center curve along which the contour is being swept

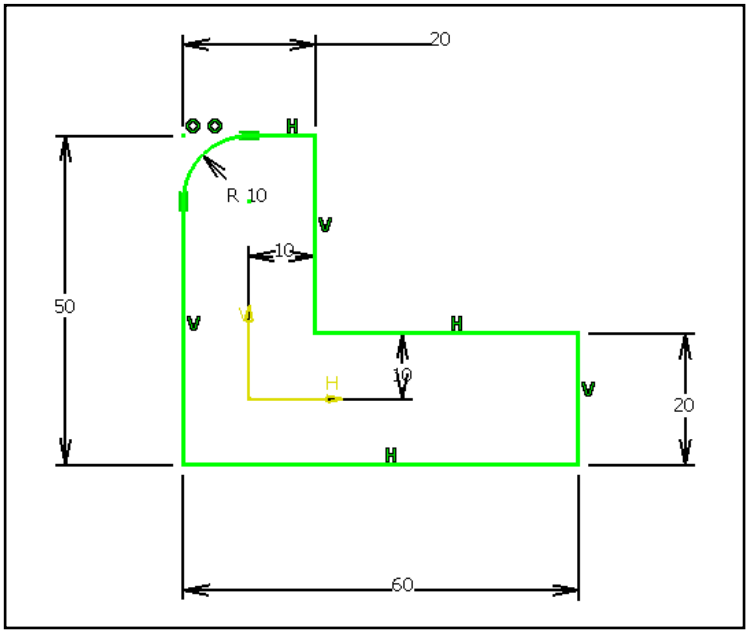


Figure 45: Sketch of the *Profile*

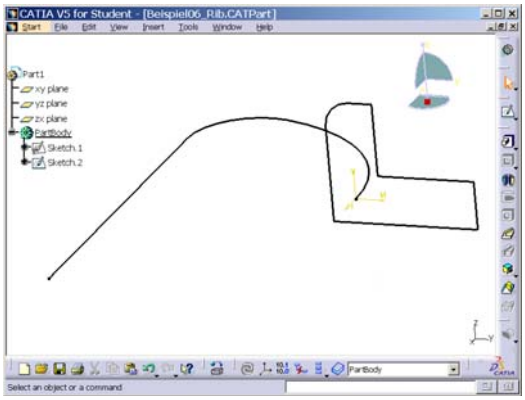


Figure 44: *Profile* and *Center curve*

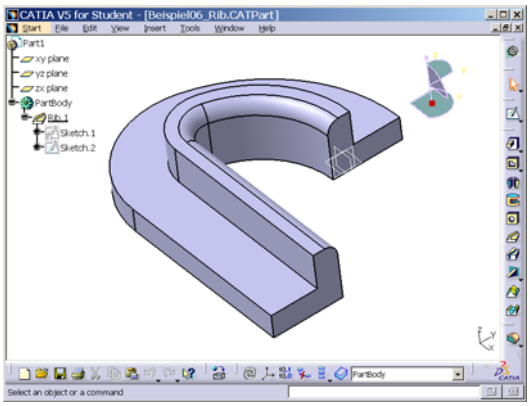


Figure 43: Rib

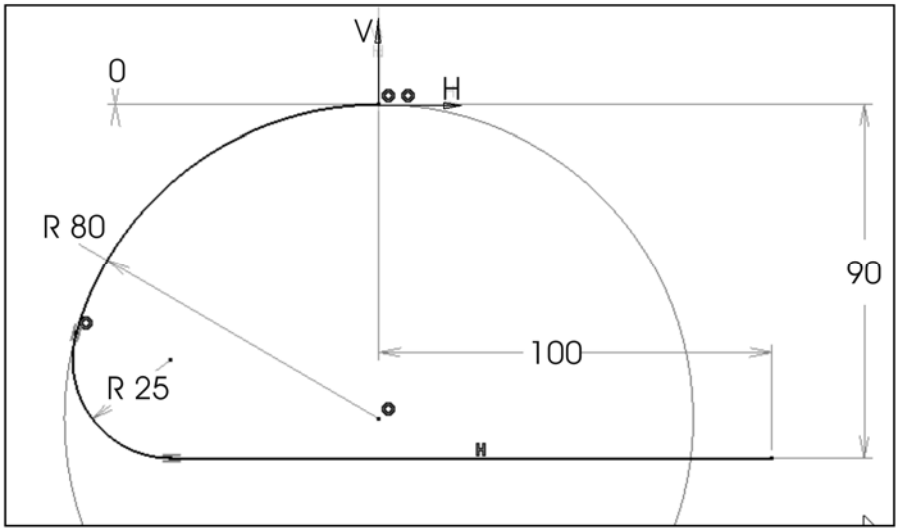


Figure 46: Sketch of the *Guide curve*

After creating both sketches in two perpendicular planes, the feature *Rib* can be selected.

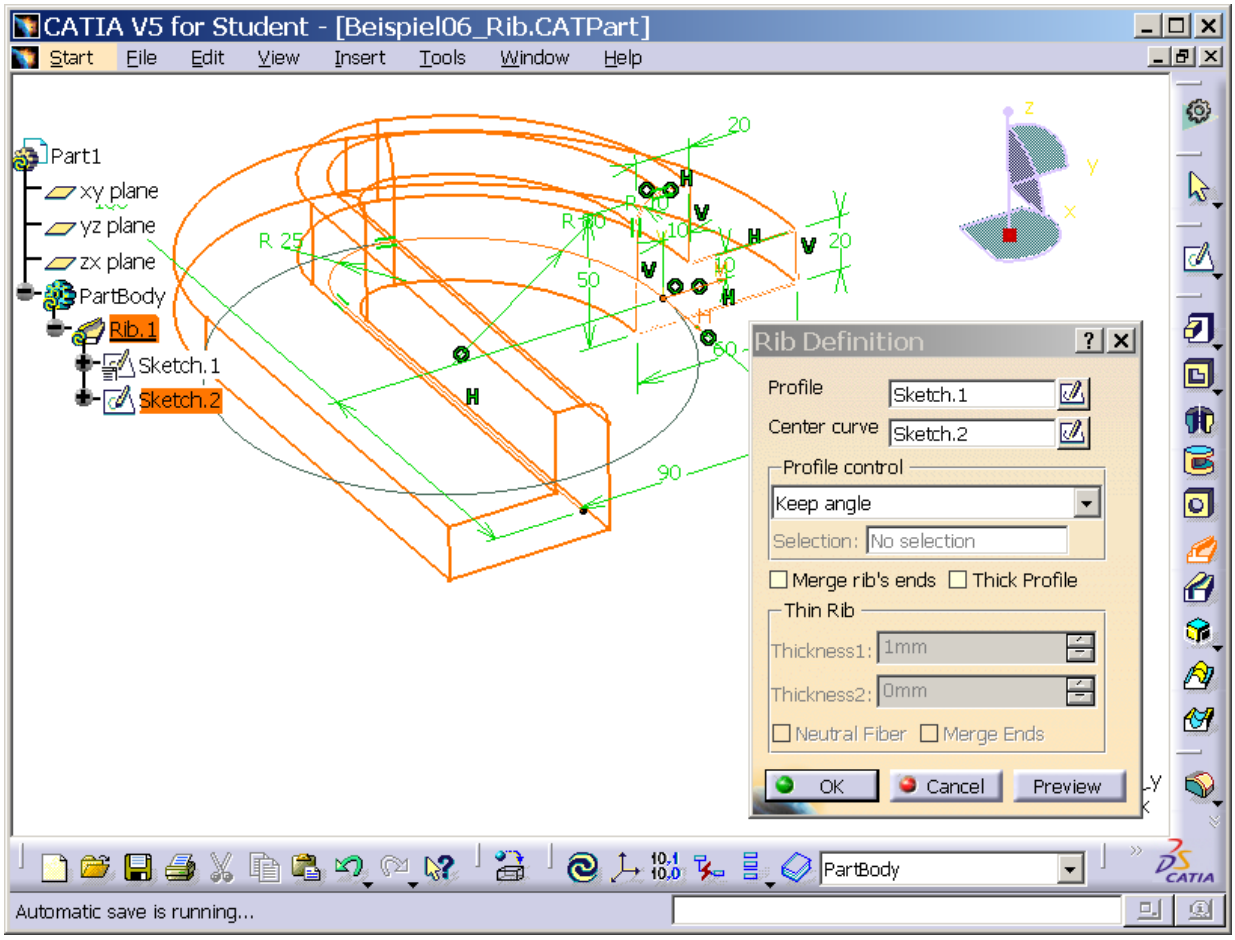


Figure 47: Defining a*Rib*

The definition box prompts a *Profile* and a *Guide Curve*.

☺ **Annotation**: The sketch support can be changed: The sketch has to be clicked with the right mouse button and *Sketch.x object / Change Sketch Support* has to be selected. The new support plane has to be picked. The existing constraints have to be adapted, if they refer to former reference elements.

6.2 Manipulation features

The following functions are a selection of the toolbar manipulation features and can be used to modify bodies.

Pocket

Groove

Hole

The manipulation features can, similar to the 3D basic features, base on sketches created in the sketch mode. Only the function *Hole* allows the definition of parameters within the dialog box; the positioning of the hole is done with a sketch that can be activated within the box.

Example 7: Plate

Intention: Application of the features Pocket, Groove, Hole

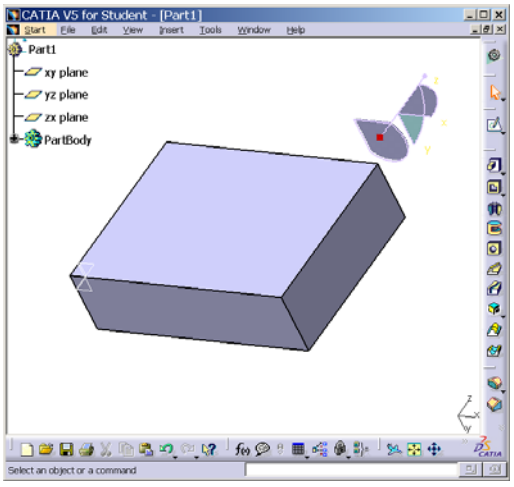


Figure 48: The basis solid

As a basis solid, a *Pad* measuring 80 x 110 x 30mm has to be generated.

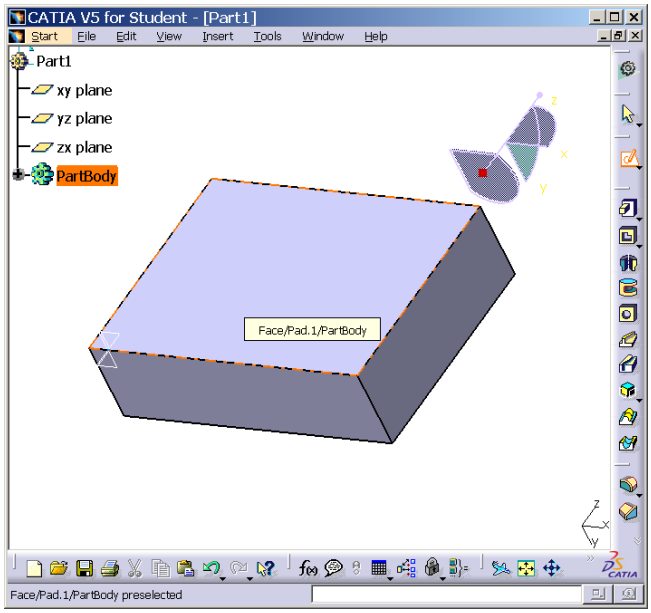


Figure 49: Selecting a face of the pad as sketch support

The Feature *Pocket*

Pocket creates a 'negative pad', i.e. it is removed from the basis solid. To create this negative pad, a profile is required. Therefore a surface of the solid pad is selected as sketch reference for the contour.

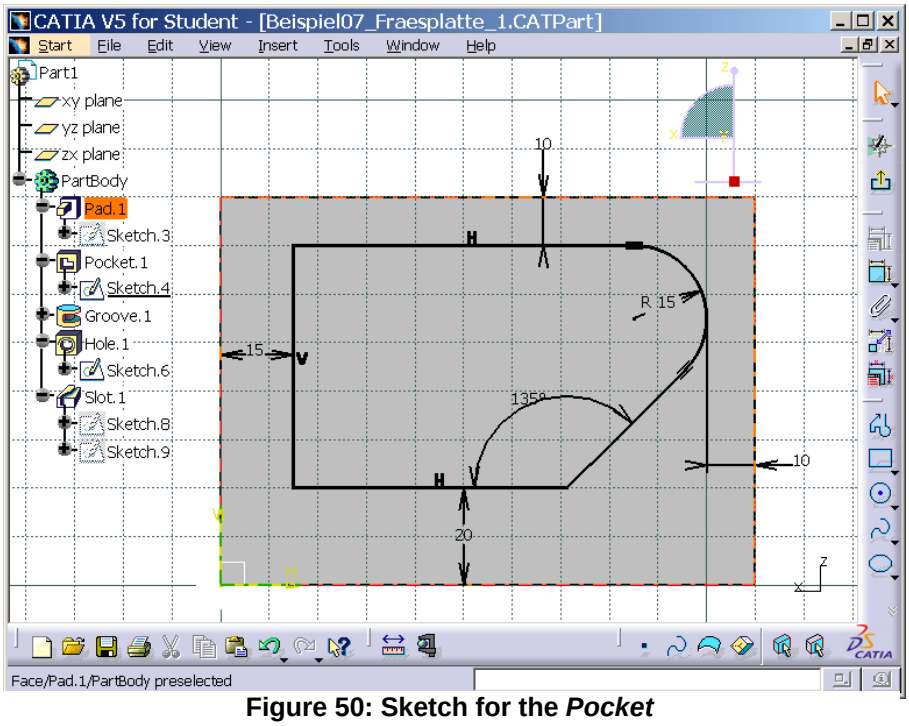


Figure 50: Sketch for the *Pocket*

All geometrical elements should be constrained clearly and without ambiguity.

☺ **Annotation:**
It is valid to use edges, wireframe elements or other elements outside the sketch as references for the sketch. Figure 50 shows the sketch using the edges of the pad as references for different dimensions.

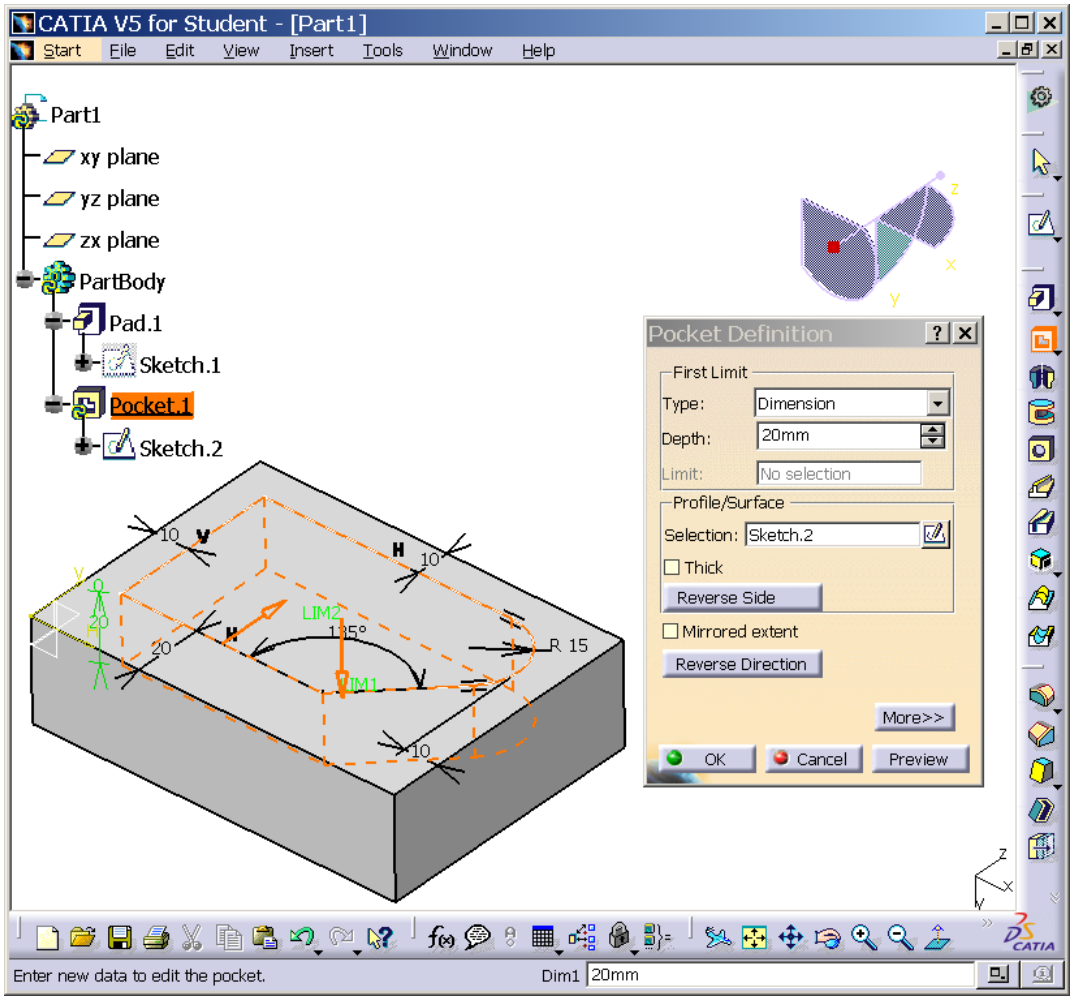


Figure 51: *Pocket* definition

After picking the button *Pocket*, a definition box appears. The *Depth* is set to 20mm, the *Profile* is the sketch created before.

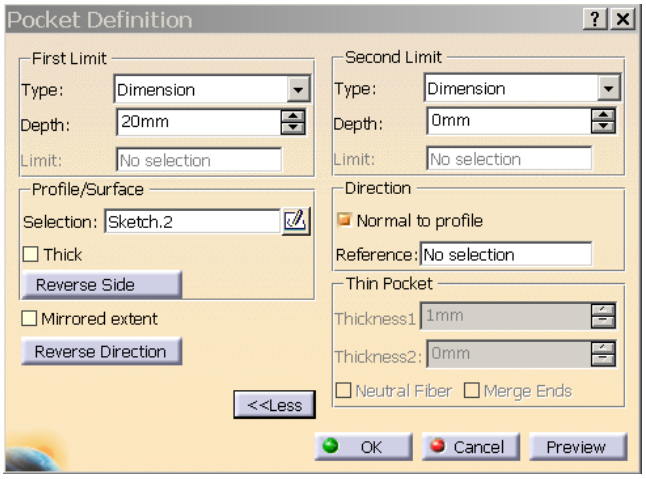


Figure 52: Extended *Pocket* definition box

Selecting *More* opens an extended *Pocket* definition box providing detailed options for limitations, shape and extension of the pocket.

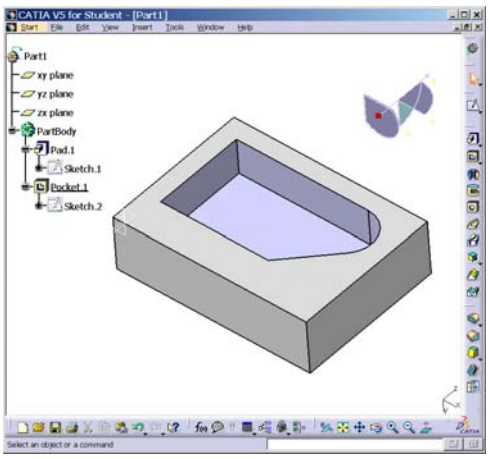


Figure 53: Body including *Pad* and *Pocket*

Figure 53 shows the basic pad and the cut, displayed in shaded view. The specification tree contains the *Pad* and the according *Sketch.1*. Underneath, the *Pocket* and *Sketch.2,* that it is

based on, are displayed. Any modification of the *Pocket* can be carried out by simply selecting a face of the *Pocket*, or by selecting the *Pocket* within the specification tree. A definition window of *Pocket.1* appears and modifications can be performed.

The Feature *Groove*

A *Groove* is a negative *Shaft*, the definition happens similar to the function *Shaft*. The input parameters have to be a *Profile* and a *Rotation Axis*.

As reference plane of the *Profile* sketch a side face of the basic *Pad* is selected.

The definition box prompts the *Profile* and the *Axis*.

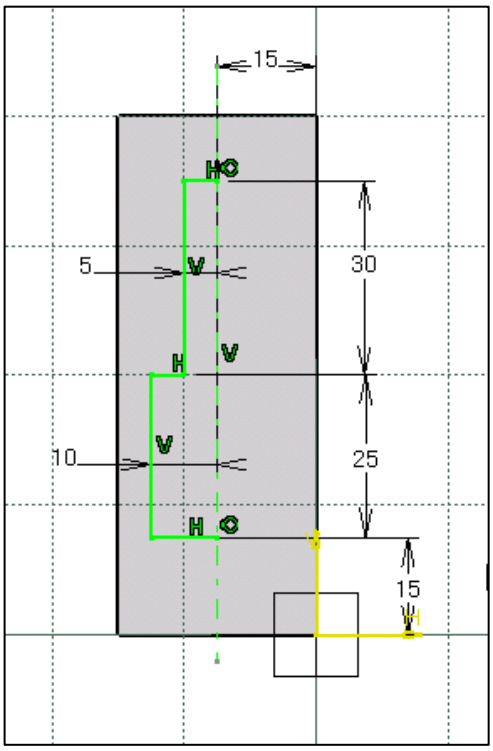


Figure 54: Sketch of the *Profile*

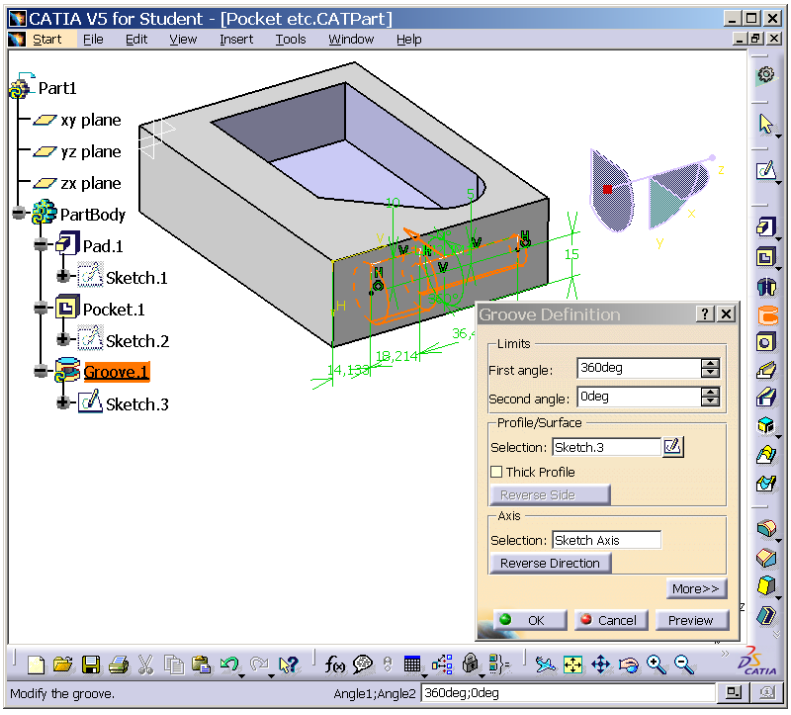


Figure 55: *Groove* definition

Figure 56 shows the complete body including the removed *Groove*.

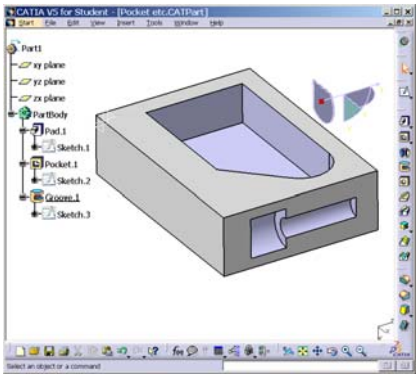


Figure 56: *Pad* with *Pocket* and *Groove*

The feature *Hole*

Holes and threads can be created on existing bodies using the feature *Hole*.

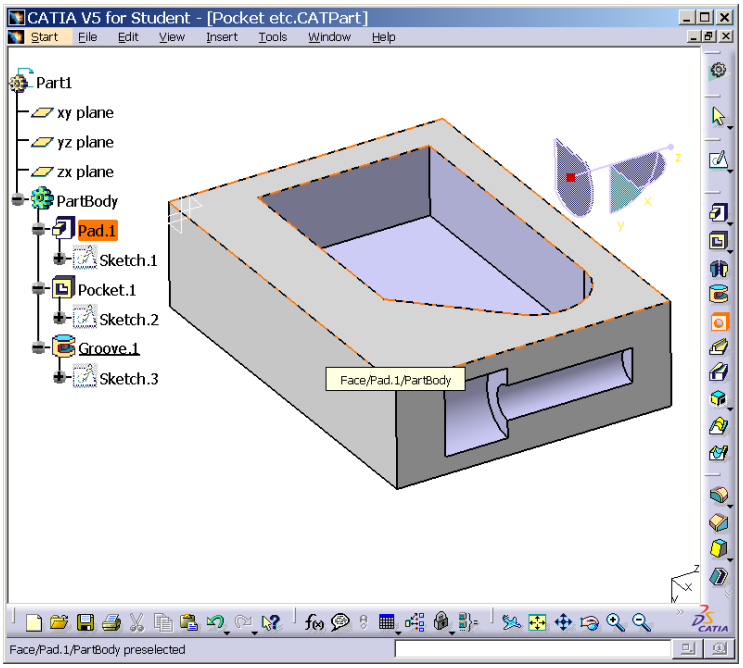


Figure 57: Selecting the reference plane for the *Hole*

As an example, a *Hole* is applied on the upper face of the *Pad*. Therefore the icon *Hole* is activated and the according surface has to be picked. All relevant input data are can be defined in the dialog box. The *Positioning Sketch* controls the position of the *Hole.*

The desired *Hole* measures 10mm (*Diameter*) x 10mm (*Depth*). The drill ought to be an ordinary one (*V-Bottom*) with an apical angle of 120°. The *Hole* ought to be perpendicular to the surface.

Other options for defining the *Hole* are shown subsequently.

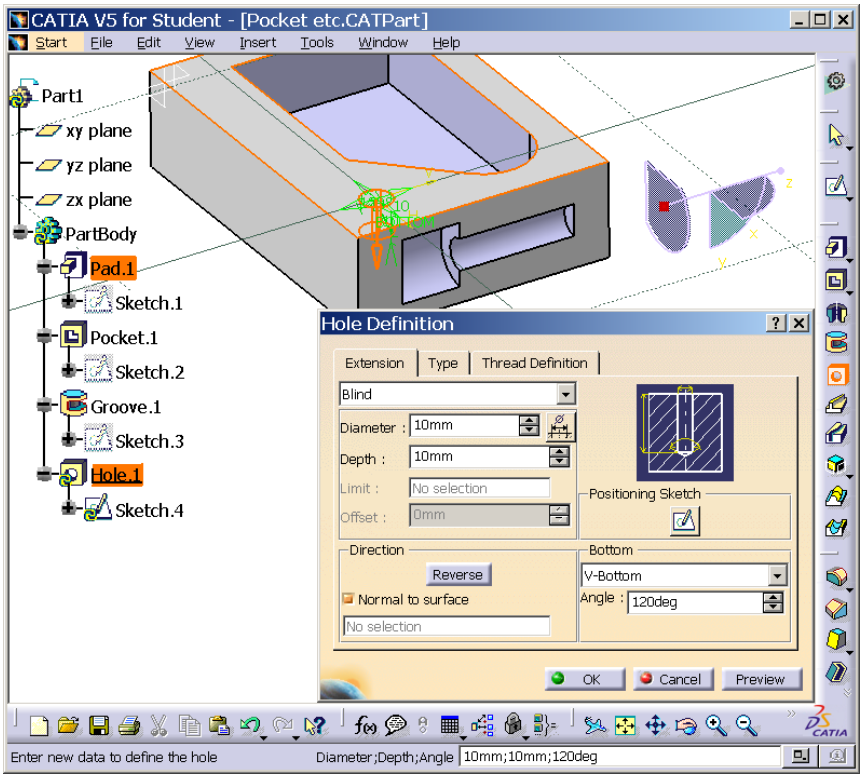
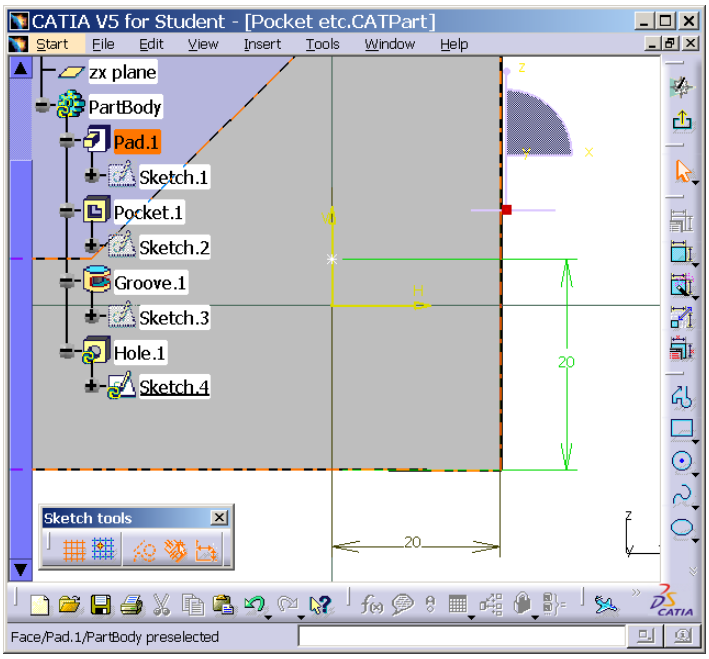


Figure 58: *Hole* definition



The *Positioning Sketch* of the *Hole* enables the definition of the *Hole* location on the reference surface.

CATIA positions the coordinate system of the sketch at the place that has been clicked at when defining the reference surface for the hole. If that point is already constrained (e. g. the point is already fixed) a *Positioning Sketch* is not needed.

Figure 59: *Positioning Sketch* for the *Hole*

Figure 60 shows the complete plate, including the *Hole*. In case of changeing parameters, the definition box can be accessed by double clicking the *Hole*.

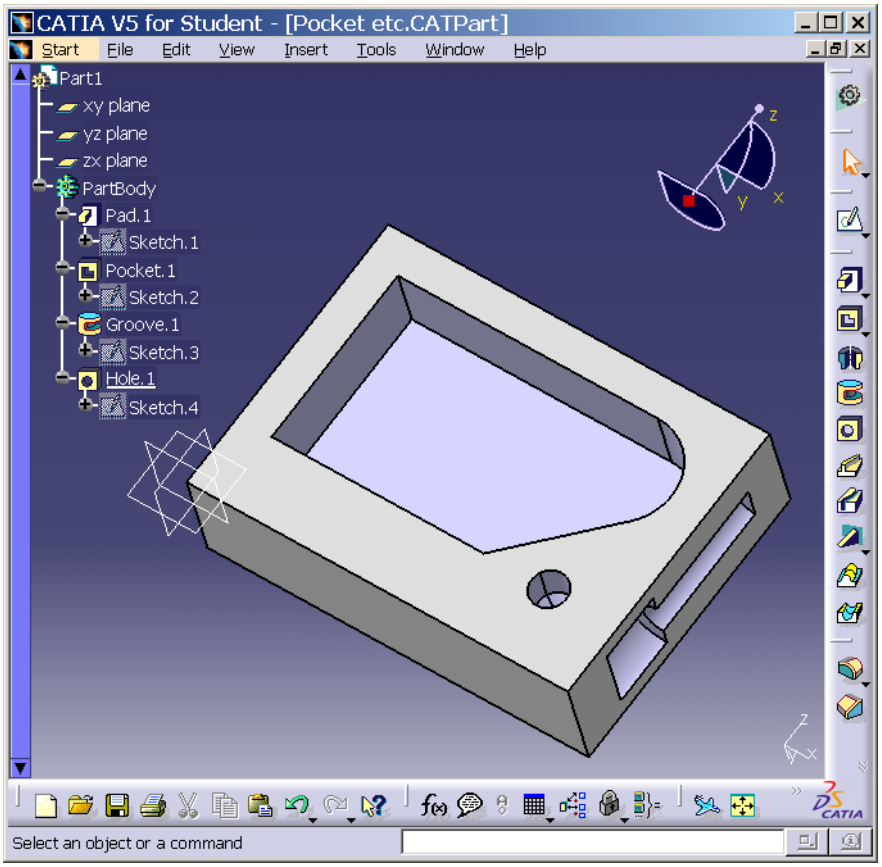


Figure 60: Plate with *Pocket, Groove* and *Hole*

The following options are available in the definition window *Hole:*

Extension:

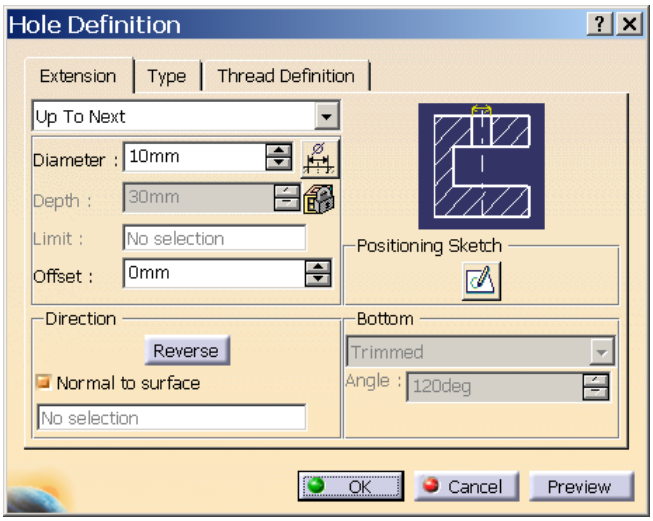


Figure 61: *Hole Definition, Extension*

The following settings are available:

- *Type of Relimitation of Hole Limit* (*Blind, Up to Next, Up to Last, Up to Plane, Up to Surface*)
- Dimensions of the *Hole*.
  Direction (normal to the surface or along an axis)
- Location of the *Hole*, defined by a *Positioning Sketch.*
- *Bottom* of the *Hole* (*Flat* or *V-Bottom*)

Type:

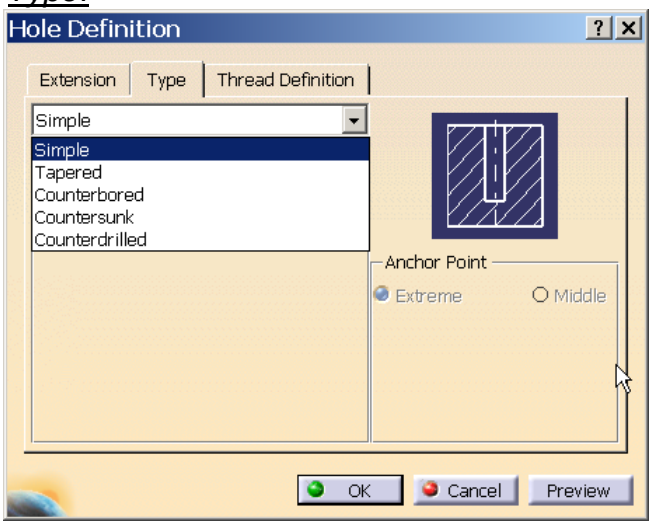


Figure 62: *Hole Definition, Type*

This menu allows the definition of a counterbore; several types are possible. If chosen, the required parameters for the type are prompted.

Thread Definition:

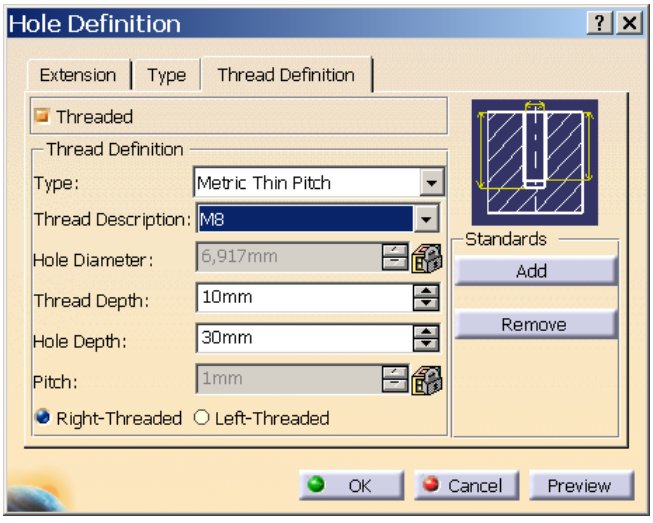


Figure 63: *Hole Definition, Thread Definition*

To create a *Thread* all parameters have to be defined using the according window. The core diameter is calculated by CATIA (in case of standard theads).
When generating a 2D *Drawing,* the *Thread* specifications in the drawing can be created by CATIA itself.

Helpful additional functions

Applying material

To apply material to the plate, the appropriate feature has to be selected. A dialog box opens, offering several material types. The element that has to be supplied with material has to be marked, as the material has to be. By selecting *Apply material*, the material properties are assigned to the geometrical element, and the icon is displayed in the *Specification Tree*.

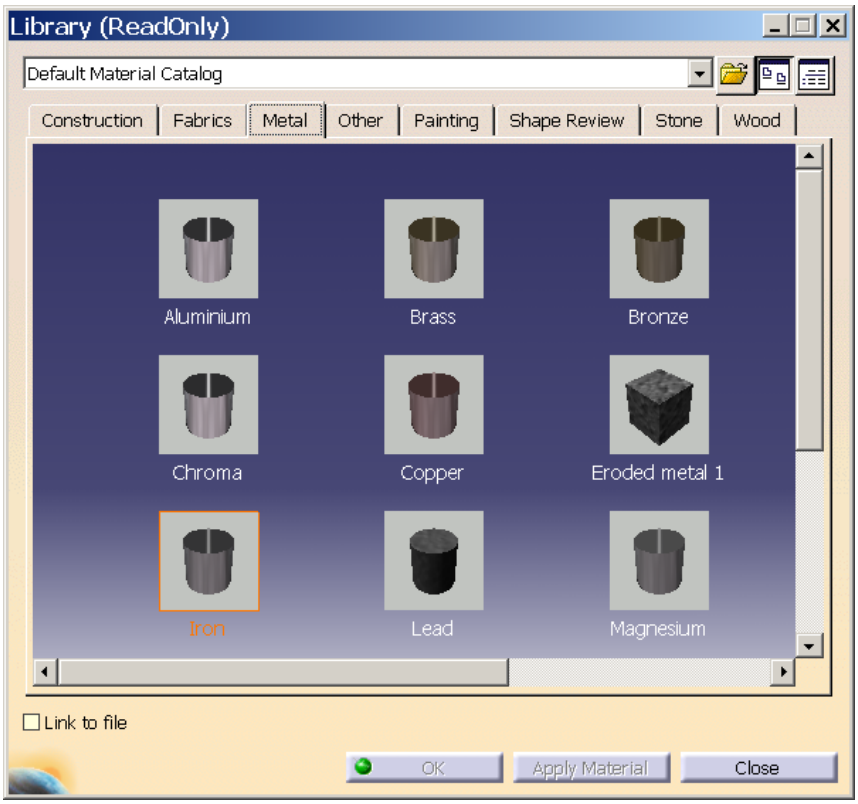


Figure 64: The dialog box for applying material

Measure Inertia

Beside the common measure function for values and distances, the icon *Measure Inertia* activates a number of measurings, such as volume, mass, area or inertia using the set material properties. Several options can be set, e.g. an axis or a coordinate system for the inertia calculation, can be selected.

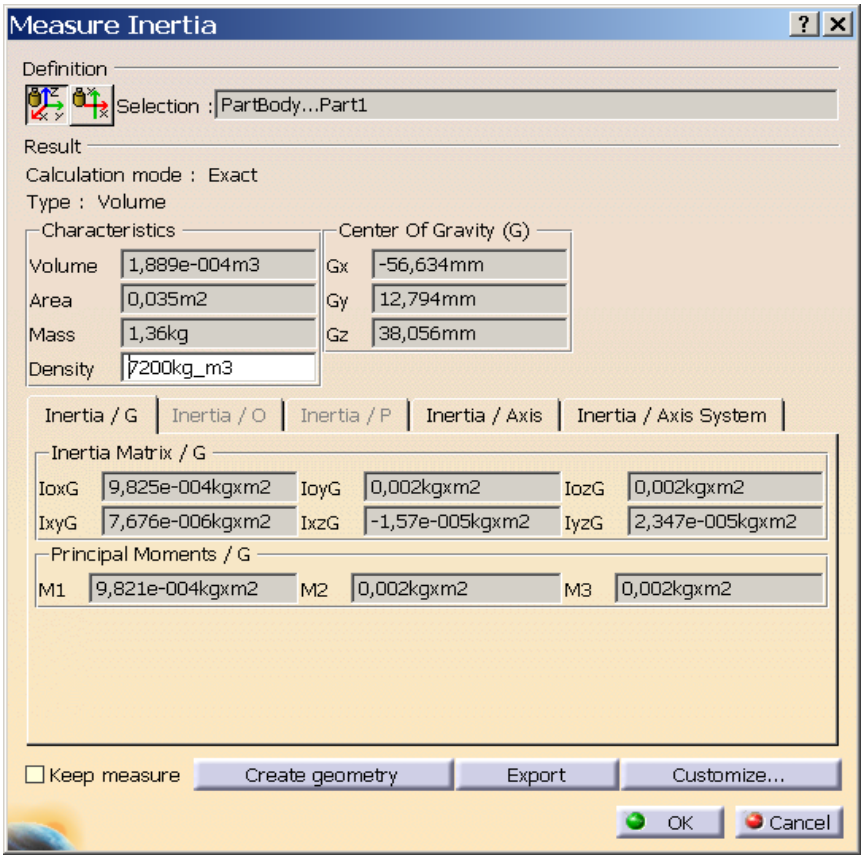


Figure 65: Measure Inertia

Using *Search*

The feature *Edit / Search* can be useful when looking for specific elements. There are several properties to search for, such as *Color, Type* or *Name*. When using *Name*, it is possible to search for elements other than geometrical items, e.g. *Constraints*.

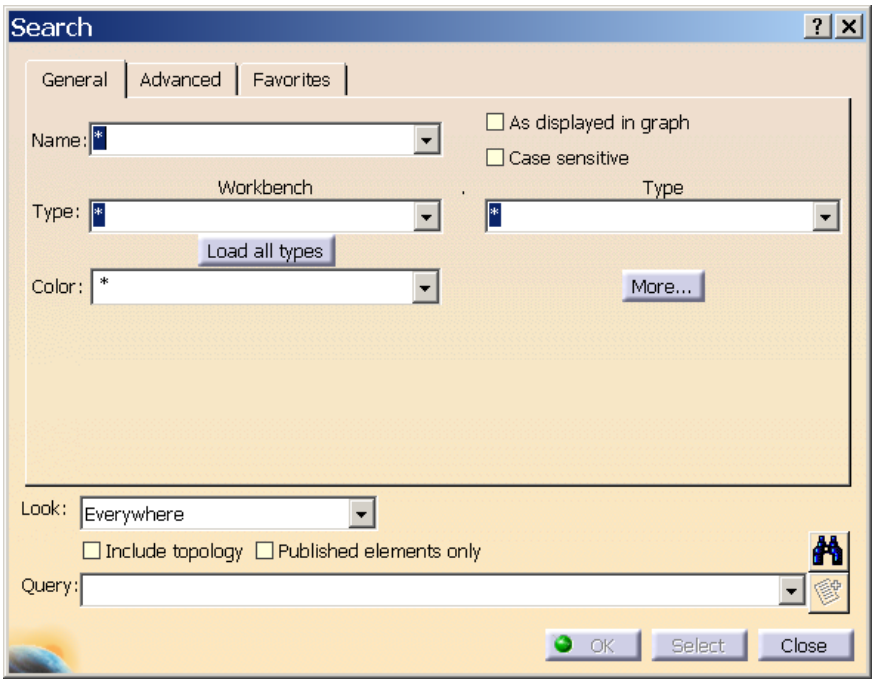


Figure 66: The *Search* dialog box

Taking pictures of elements

CATIA offers a feature to take pictures of 3D models. Therefore *Tools / Image / Capture* has to be selected. A toolbar appears. The *Options* Icon opens a setup window.

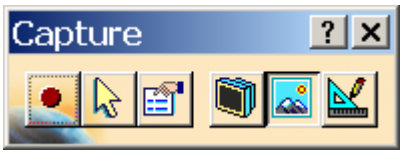


Figure 67: The *Capture* Toolbar

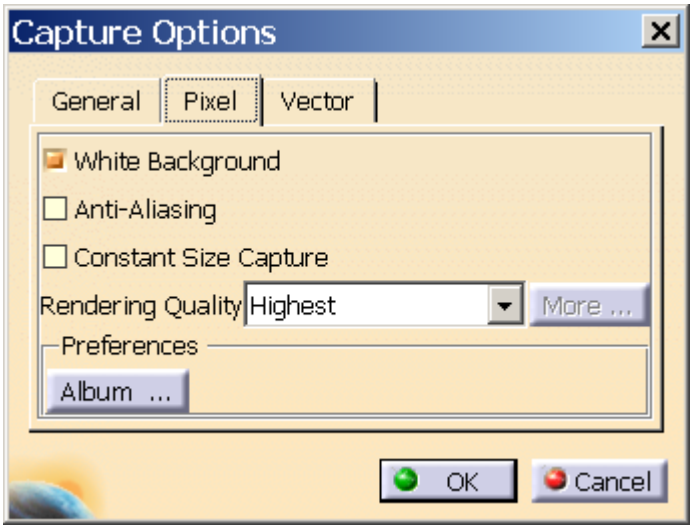


Figure 69: The setup window (*Options*)

To take the picture, the *Capture* button has to be picked. A preview of the image is shown. It can be saved by clicking the *Save as* icon.

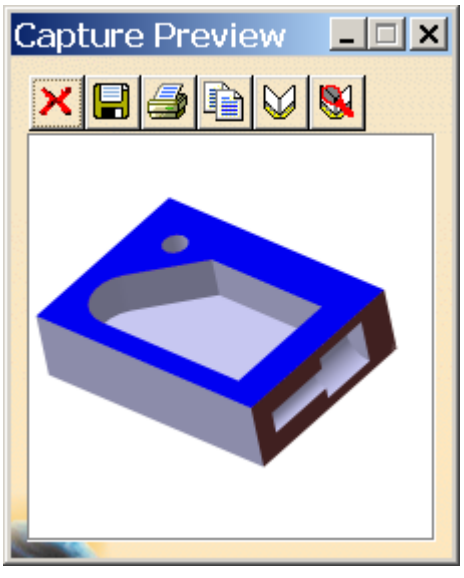


Figure 68: The *Capture Preview*

Example 8: Clevis

Intention: Design of simple solid bodies

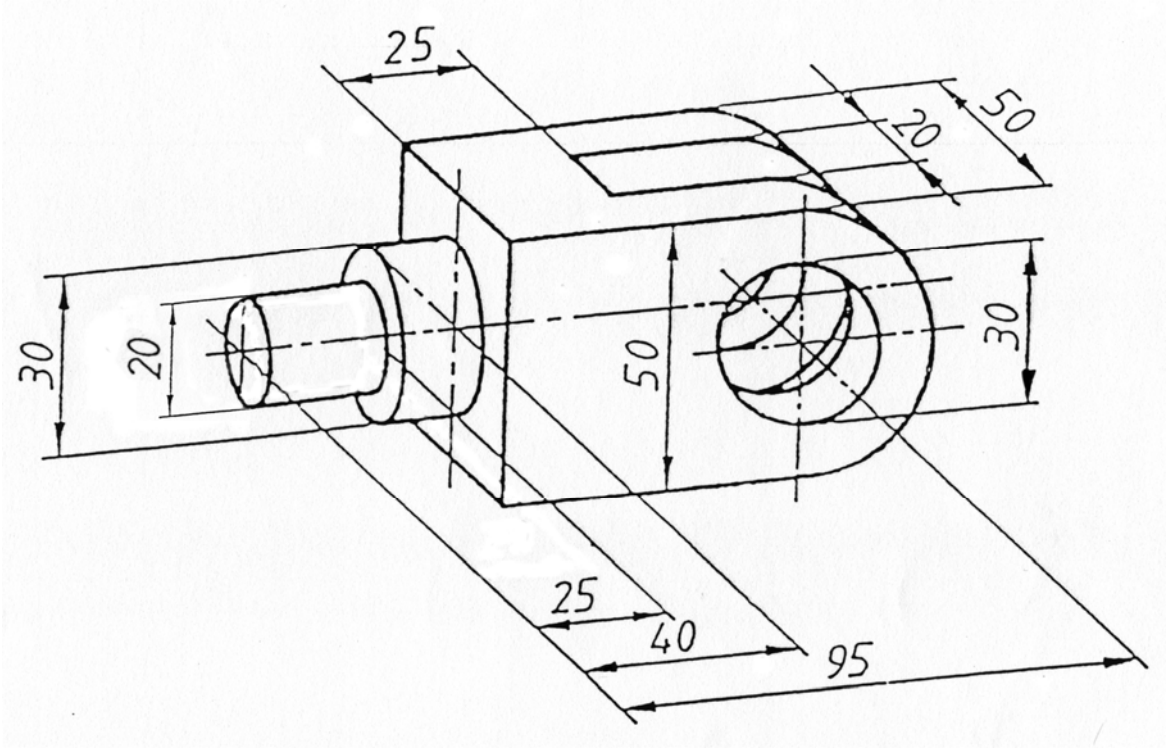


Figure 70: Clevis

Example 9: Lever

Intention: Design of simple solid bodies

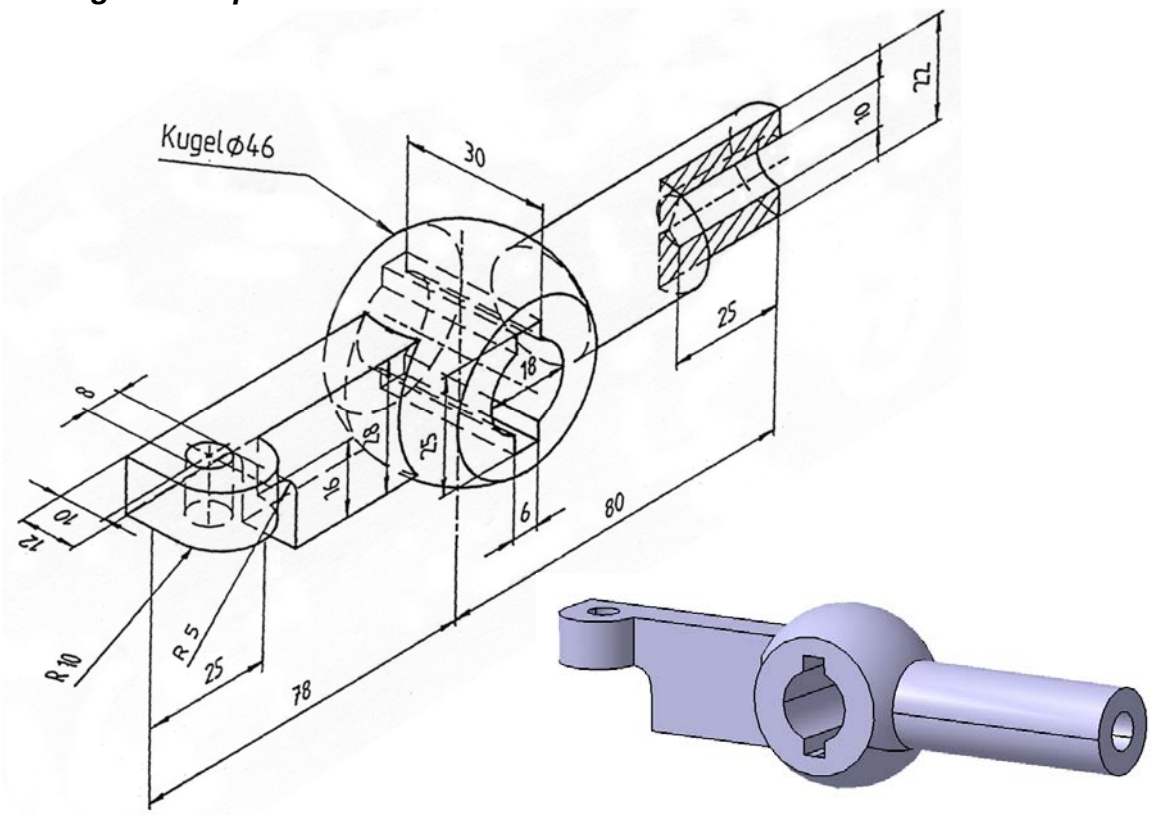


Figure 71: Lever

Example 10: Prism piece

Intention: Design of simple solid bodies

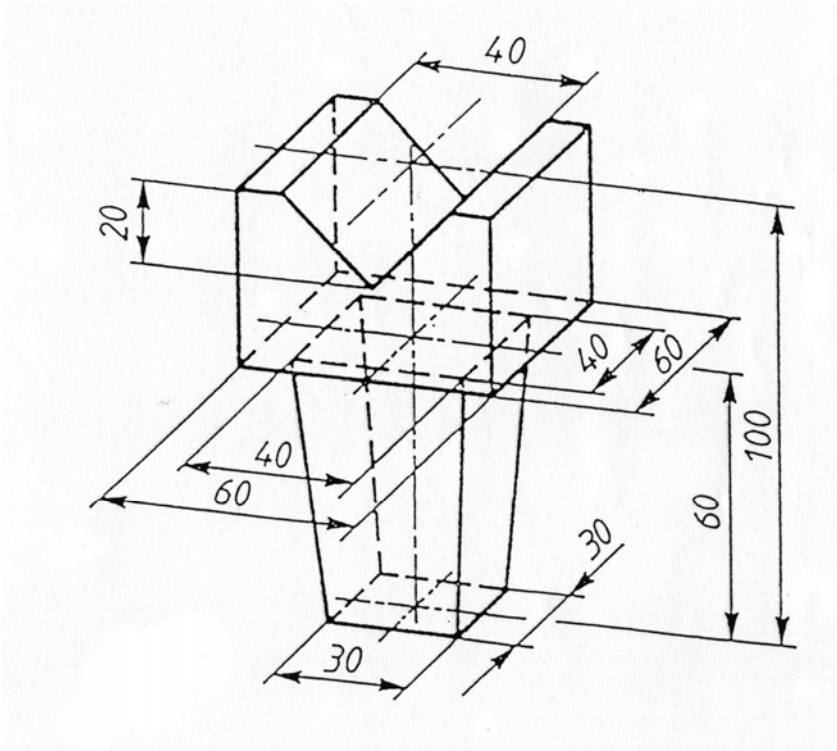


Figure 72: Prism Piece

Example 11: Sleeve

Intention: Design of simple solid bodies

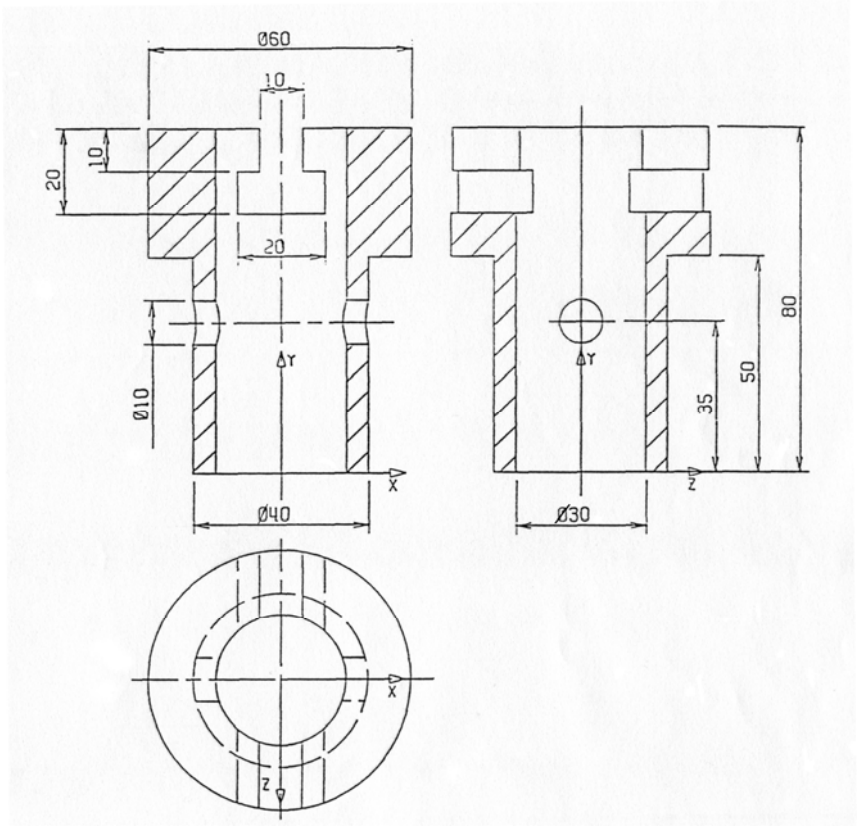


Figure 73: Sleeve

Example 12: Prism with threaded holes

Intention: Design of simple solid bodies

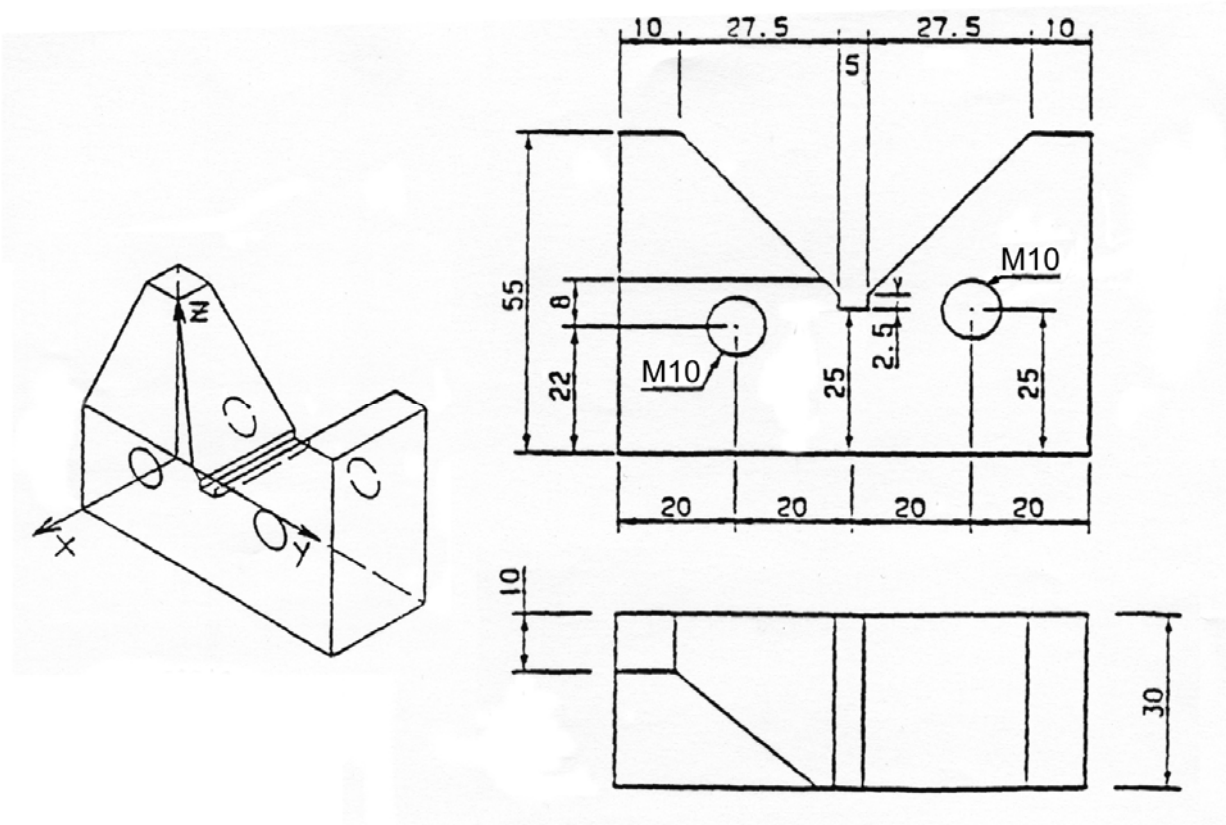


Figure 74: Prism with threaded holes

6.3. Dress-Up Features

The *Dress-Up Features* are used to implement constructive modifications on existing solids, such as *Chamfers, Fillets* or *Draft Angles*.
By means of an example these features are explained in detail.

Example 13: Angle piece

Intention: Using the Dress-Up Features

Two *Pads,* rectangular to each other positioned, serve as a basis for this example. The first component measures 100 x 80 x 20 mm and is placed in the yz-plane. The second component measures 60 x 60 x 20 mm and is positioned in the xz-plane.

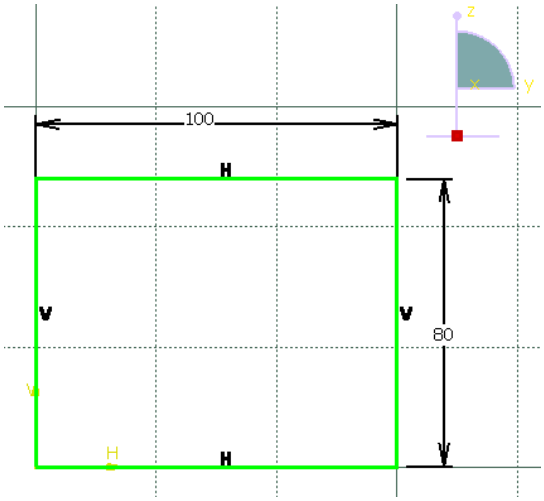


Figure 76: Sketch of the first *Pad*

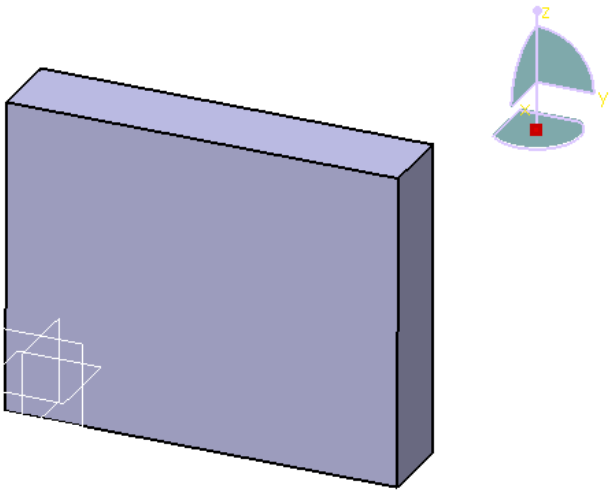


Figure 75: First *Pad*

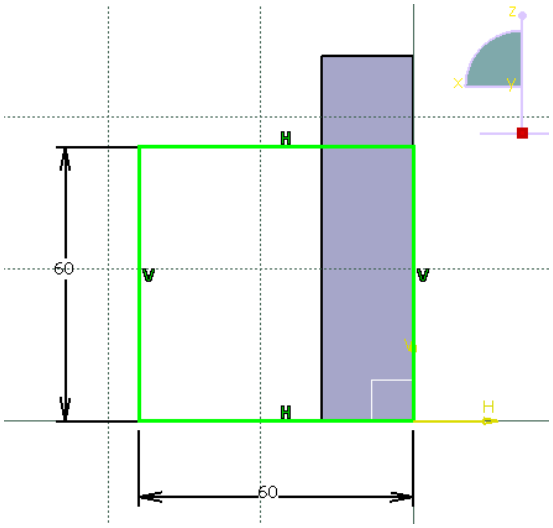


Figure 78: Sketch of the second *Pad*

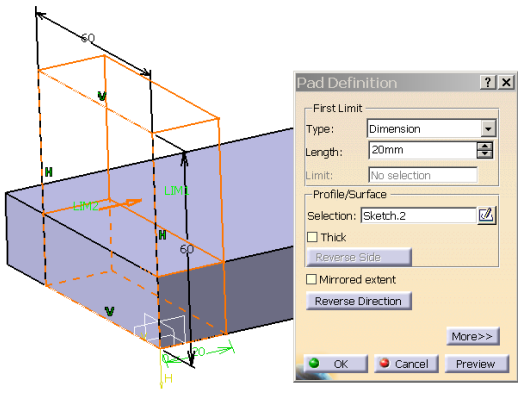


Figure 77: Creation of the second *Pad*

The feature Fillet

After having selected the button *Edge Fillet,* a dialog box specifies the fillet, prompting the *Radius*, the *Propagation* and, if desired, *Limiting Elements* (menu *More*). It is possible to fillet more than one edge at once. When trying to fillet three or more edges that concur in an acute angle, the sequence of the filleting does indeed make a difference. It has to be stated, that the result depends on the order of the filleted edges. Generally it is better to apply the fillets with bigger radii first.

The submenu *Fillets* provides features for creating fillets with variable radii, between two faces or defined by three tangent faces.

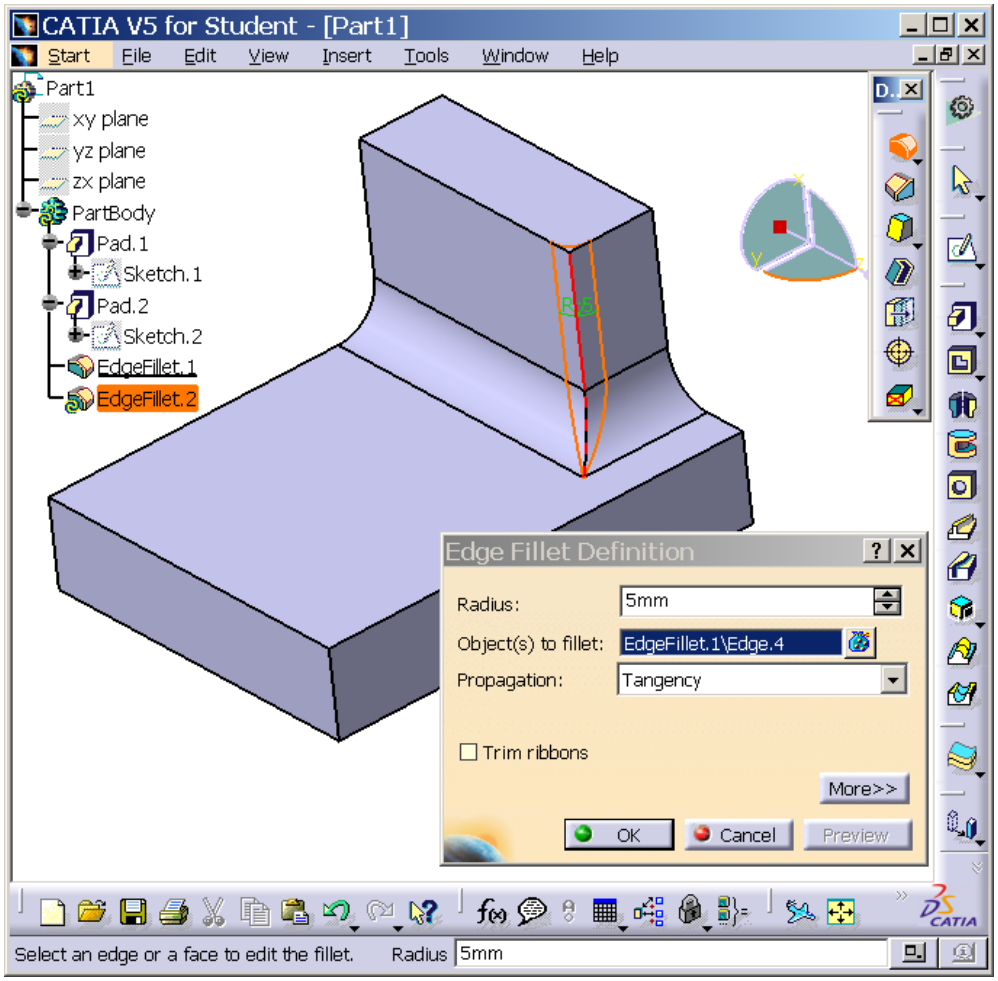


Figure 79: Creating a *Fillet*

The angle piece has to be supplied with fillets with R=10mm on the inner edges and a fillet with R=5mm on the vertical edge.

The *Specification Tree* shows the newly created *Fillets* beneath the *Pads*.

Similar *Fillets* can be generated by use of the function *Variable Radius Fillet*. The according radii have to be defined in a dialog box.

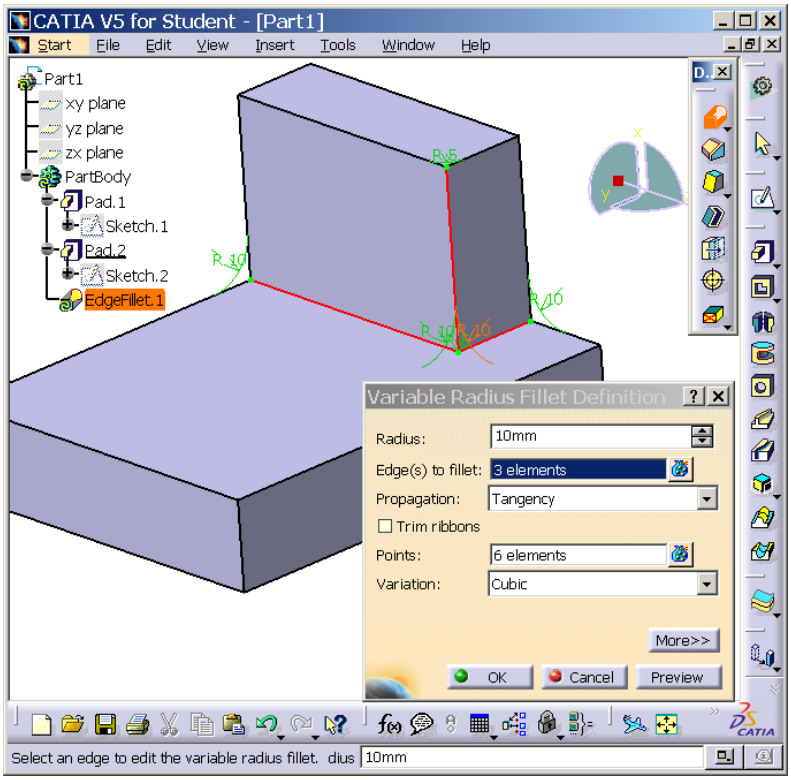


Figure 80: Defining a *Variable Radius Fillet*

To modify the *Fillet* definition, double clicked the *Fillet* (in the specification tree or on the solid in the working space) and redefine the values in the dialog box.

All single radii can be marked and modified in the definition window.

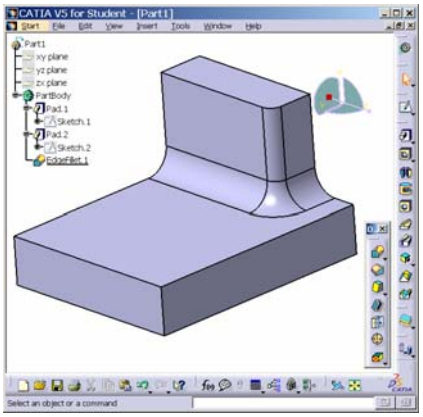


Figure 81: Filleted body

The feature *Chamfer*

Two input options are selectable for the *Chamfer* definition: - Length and Angle or
- 2 Lengths.

The *Angle* is measured from the body surface, the *Length* is measured from the original edge to the newly created edge of the chamfer. Several edges can be selected to create several chamfers in one step.

The feature *Chamfer* is inserted in the specification tree.

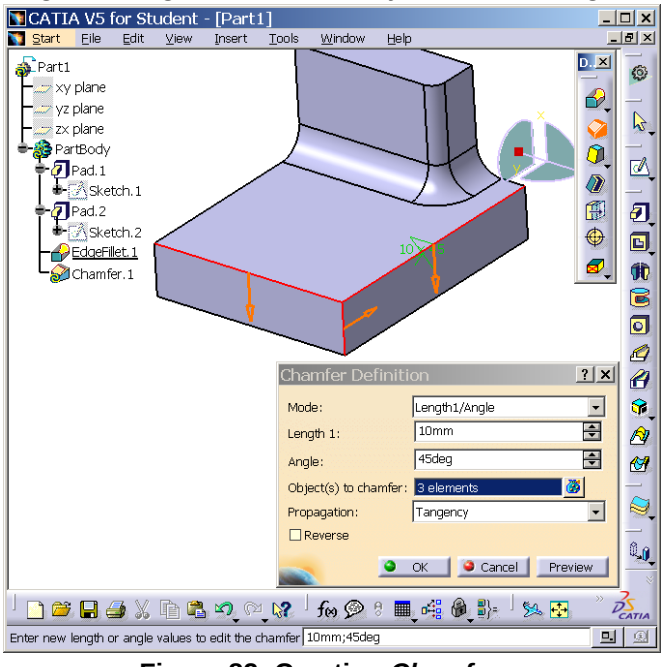


Figure 82: Creating *Chamfers*

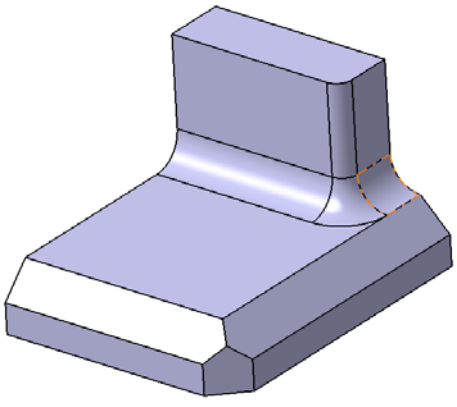


Figure 83: Part with *Chamfers*

The feature *Draft Angle*

Draft Angle is used to create slant surfaces on a solid. These drafts are needed for specific production procedures (die-casting, deep drawing, heavy stamping or others); so they are demoulded in predefined directions.

The *Draft Analysis* can check bodies concerning their demouldability.

☺ **Annotation:** It may be better to create the *Draft Angle* before filleting the body, because CATIA recognizes two filleted surfaces as one single face and applies the *Draft Angle* onto the complete surface. In the present example, delete the *Fillets* before creating *Draft* Angles.

When defining the *Draft*, the surfaces to be slanted are displayed red. The *Neutral Element* is displayed blue and represents the section where the original dimension of the body (e.g. the thickness of the body) is preserved. The demoulding direction is displayed with a red arrow.

Various parameters can be defined in the dialog box, such as *Draft Angles* and options for the *Propagation* or *Limitations*.

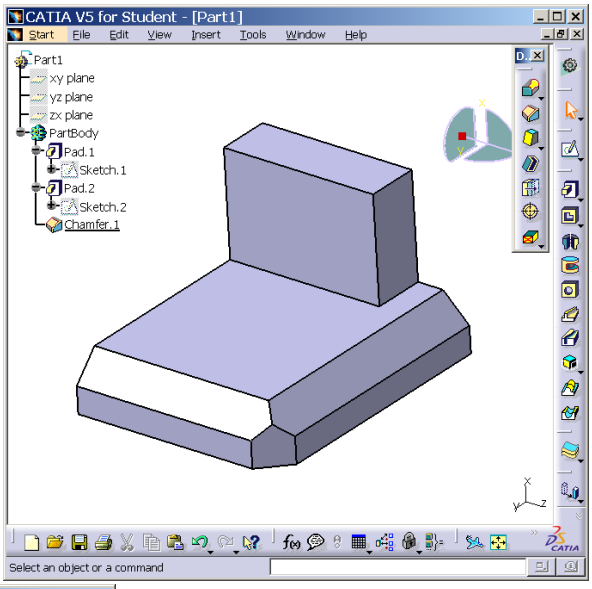


Figure 84: Part without *Fillets*

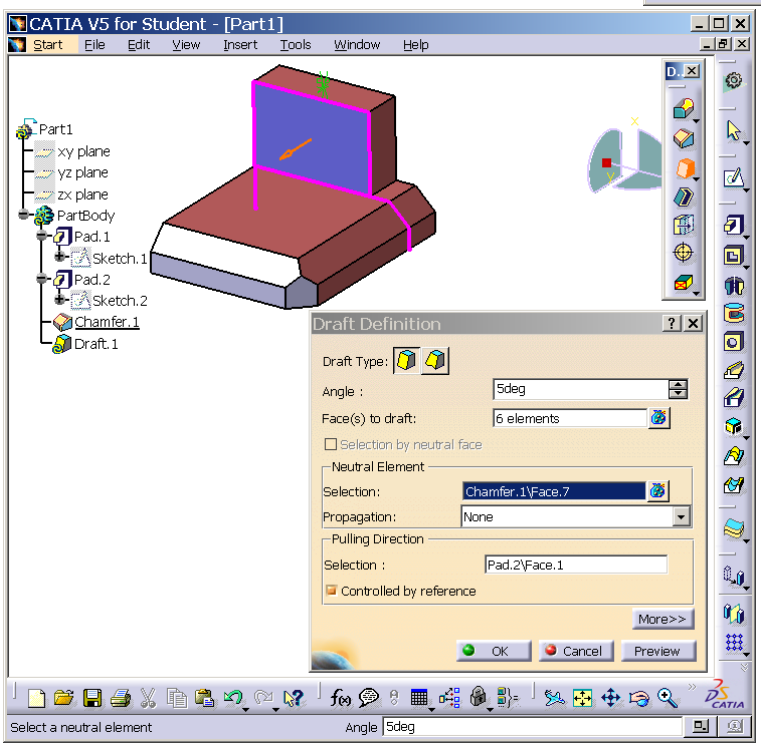


Figure 85: *Draft Definition*

The submenu *Drafts* contains miscellaneous definition options.

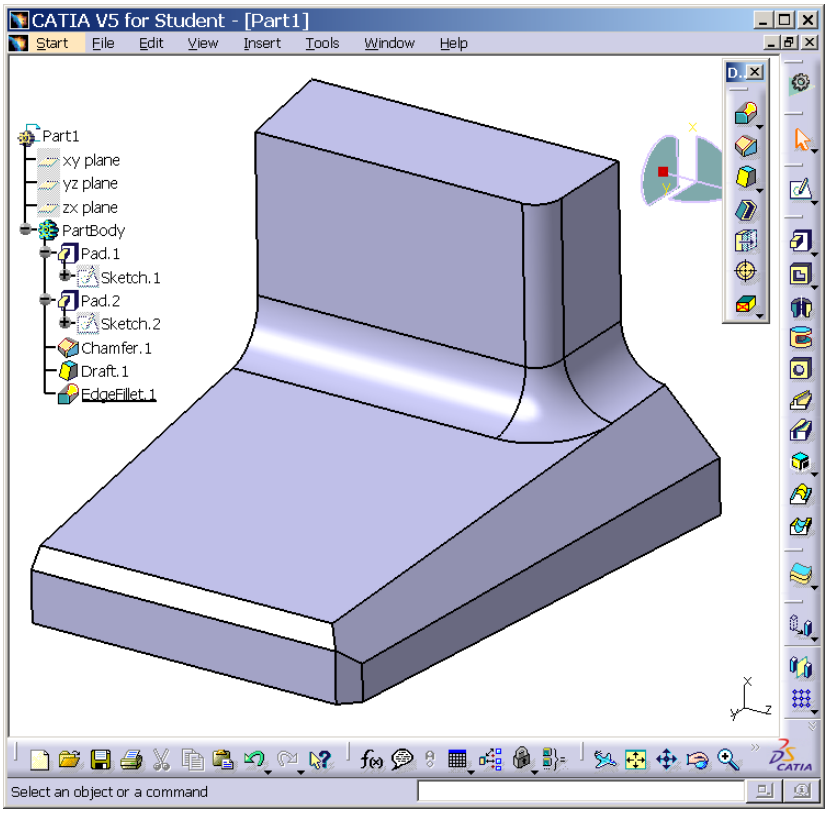


Figure 86 shows the body including the *Draft Angles* and the *Fillets*.

Figure 86: Part with *Drafts* and *Fillets*

Checking the Draft (Draft Analysis)

To check the *Part* regarding its demouldability, the *Draft analysis* is the suitable feature. The icon can be found in the *Analysis* toolbar. Before picking it, the view mode has to be switched to *Shading with Material*. The *Compass* button in the dialog box has to be selected. Now, the compass can be drawn with the mouse to a surface normal to the desired demoulding direction. Additionally, the element to analyse has to be selected. This results in a colour coding, according to the predefined settings. Figure 87 shows the

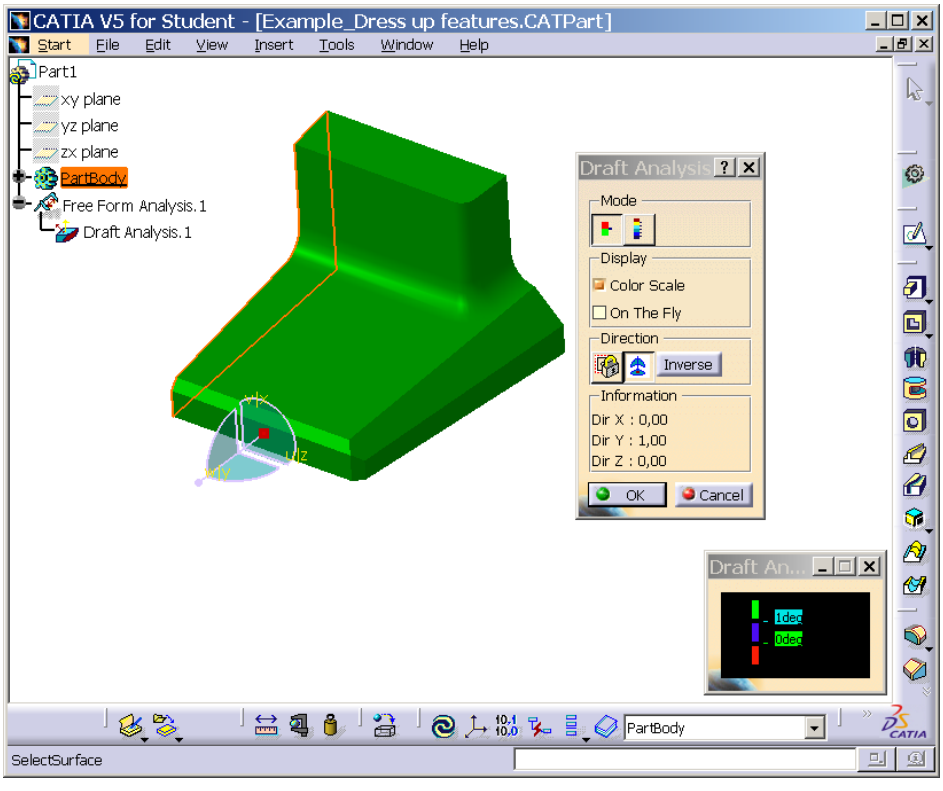


Figure 87: *Draft Analysis*

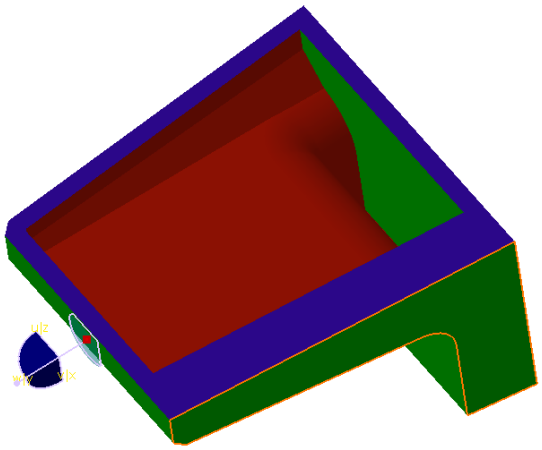


Figure 88: Surfaces that are not demouldable with respect to the given direction

analysis results of the drafted angle piece. As the check direction corresponds with the draft direction, it is demouldable. This is shown by the green colour, which identifies all surfaces that feature an angle of 1 degree or more to the demoulding direction. Surfaces with 0 degrees are displayed blue, and those that have less than 0 degrees are displayed red.

The feature *Shell*

Shell is used to hollow out bodies.

The definition of the *Shell* requires the thickness (inside and outside) and the *Faces to Remove*.

CATIA V5 generates an open shell element from the surface definition of the basic body, taking into account all geometry information.

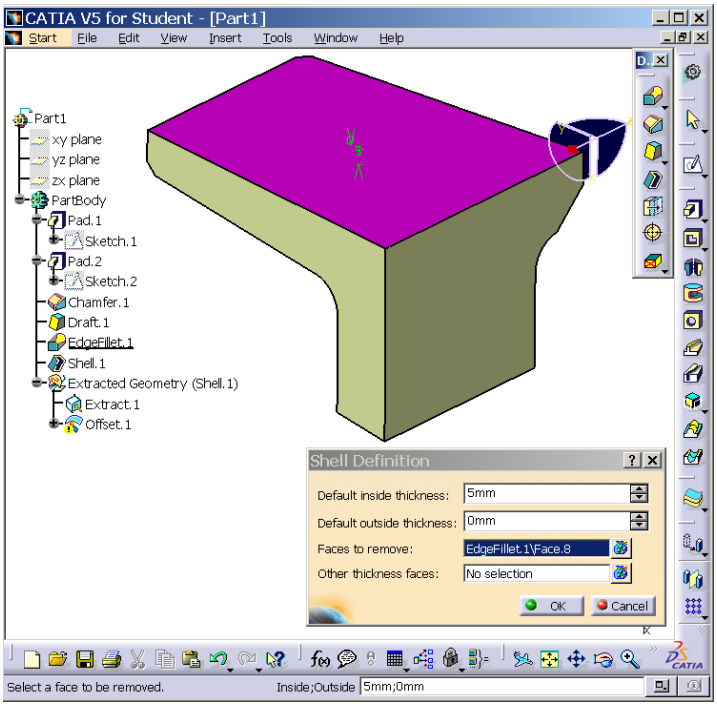


Figure 89: Defining a *Shell*

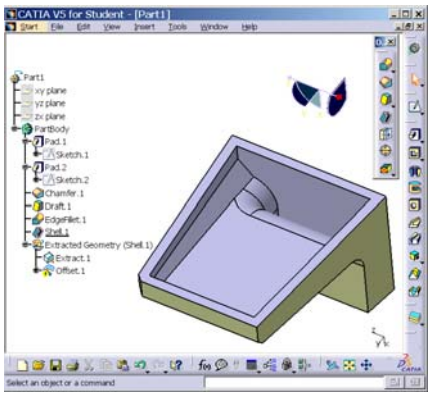


Figure 90: *Shell* element

Figure 90 shows the hollowed out body with a wall thickness of 5mm.

Feature Thickness

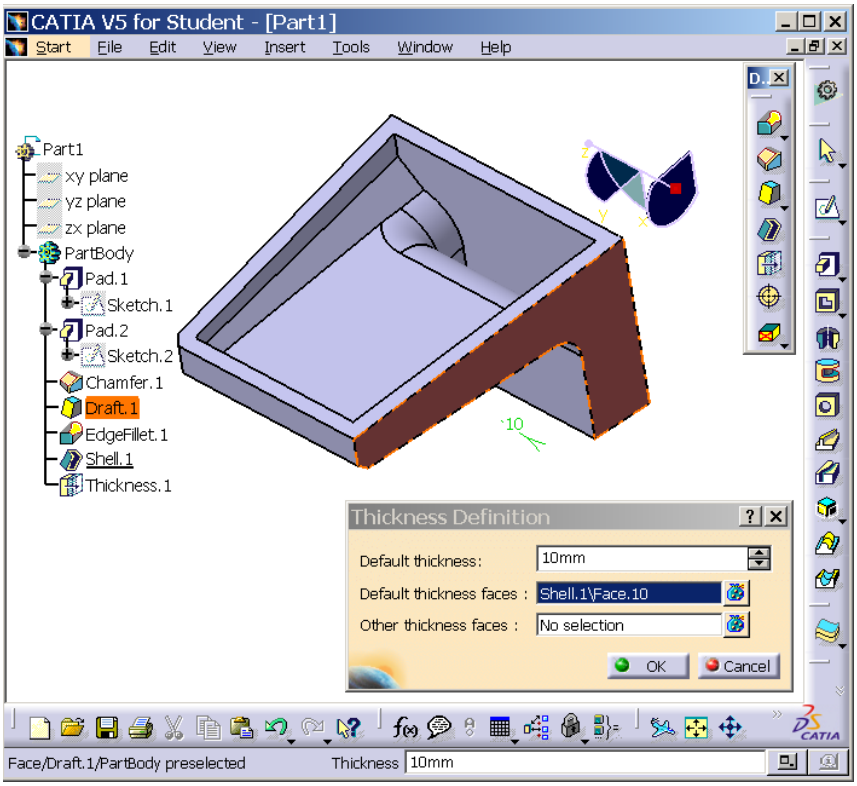


Figure 91: Applying the feature *Thickness*

Thickness adds or removes thicknesses on one (or more) faces, resulting in new boundaries parallel to the original body boundaries (e. g. for creating allowances). Different faces can be defined with different thicknesses. Modifications of dimension figures are carried out by simply clicking on them.

The features are displayed in the *Specification Tree*. They can be modified, deleted or copied in the *Specification Tree*.

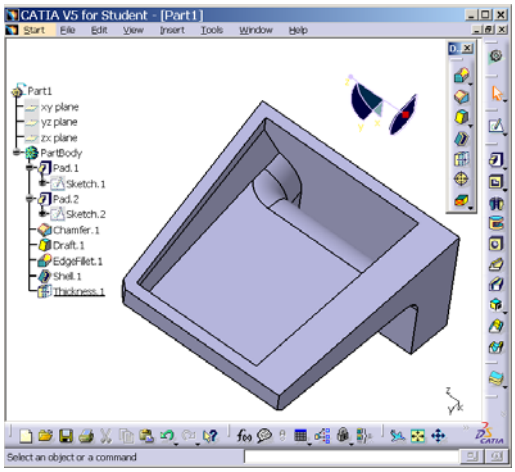


Figure 93: Part with *Thickness*

Feature *Thread*

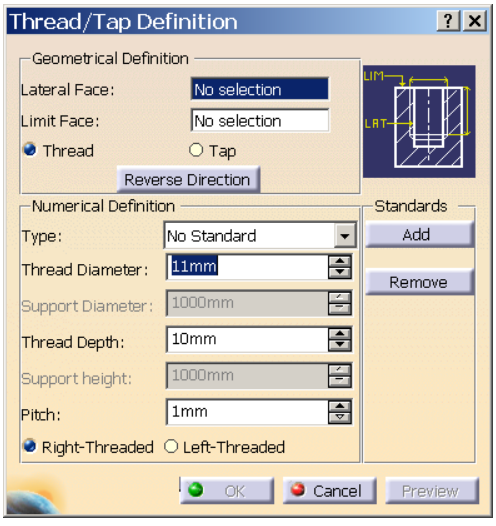


Figure 92: The *Thread* definition box

The feature *Thread / Tap* resembles the sub-feature of *Thread definition* within the *Hole* menu. Unlike the latter, this feature allows the creation of taps. The *Threads / Taps* are specified by defining the *Lateral Face, Thread Depth* and the nominal *Thread Diameter*.

Example 14: Bearing block

Intention: Application of Dress-Up Features

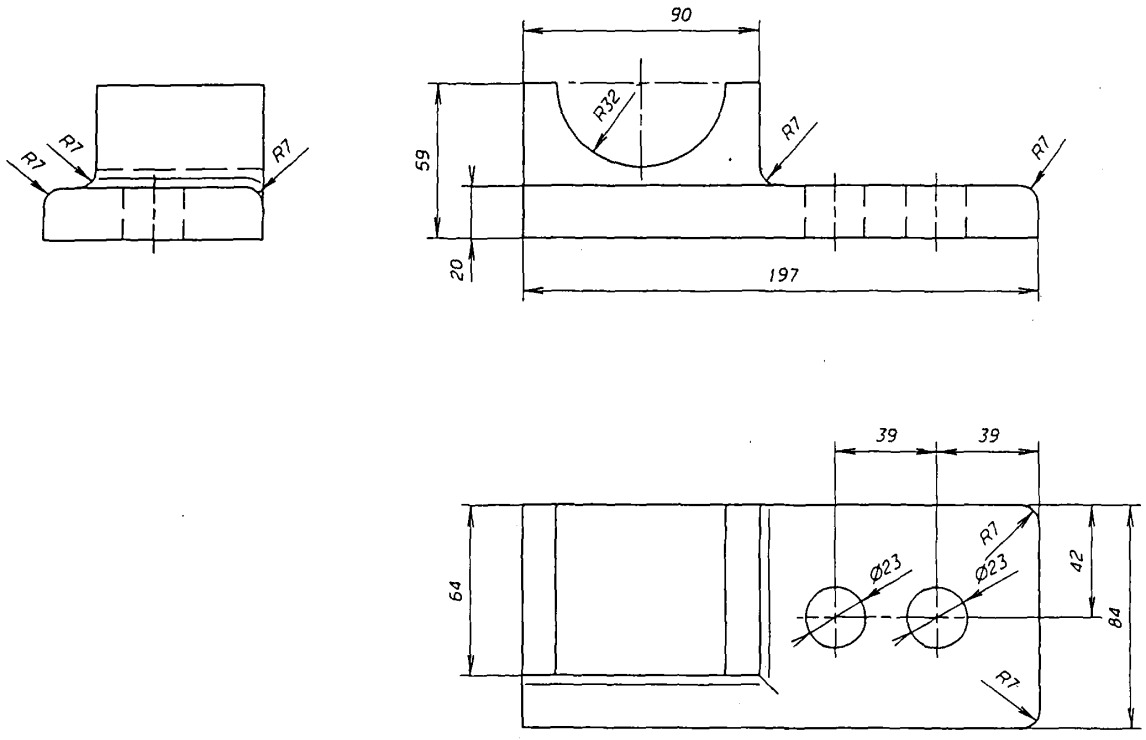


Figure 94: ***Bearing block***

Example 15: Angular prism

Intention: Part Design

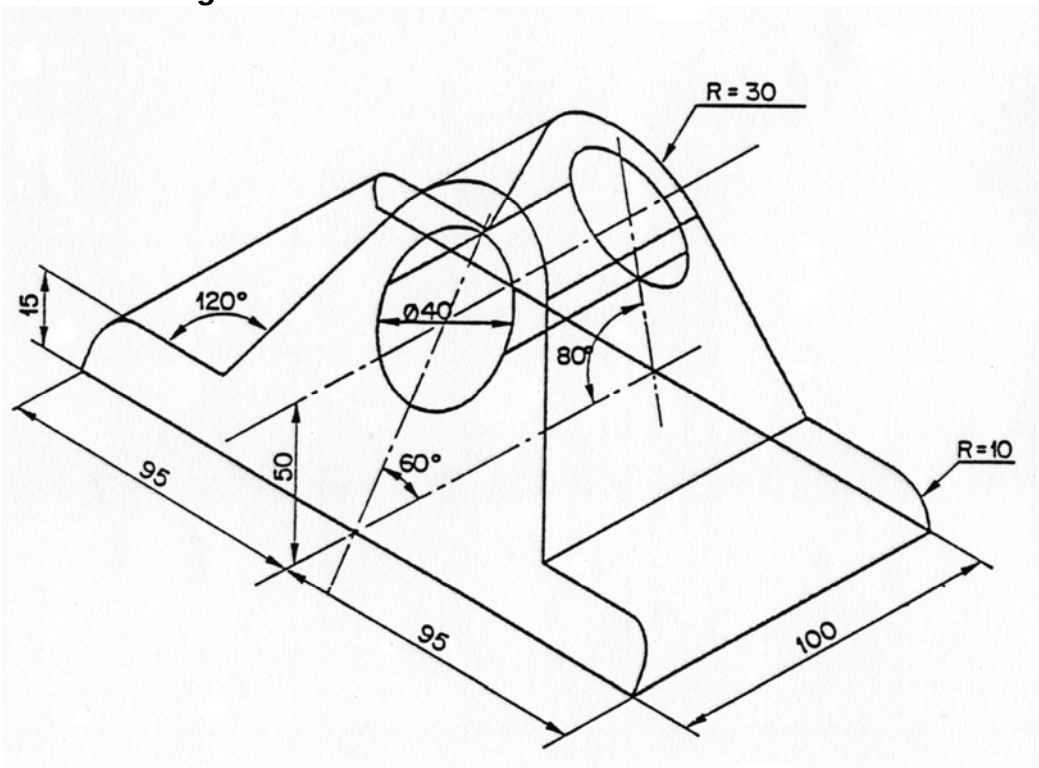


Figure 95: ***Angular prism***

Example 16: Angle anchor plate with holes

Intention: Part Design

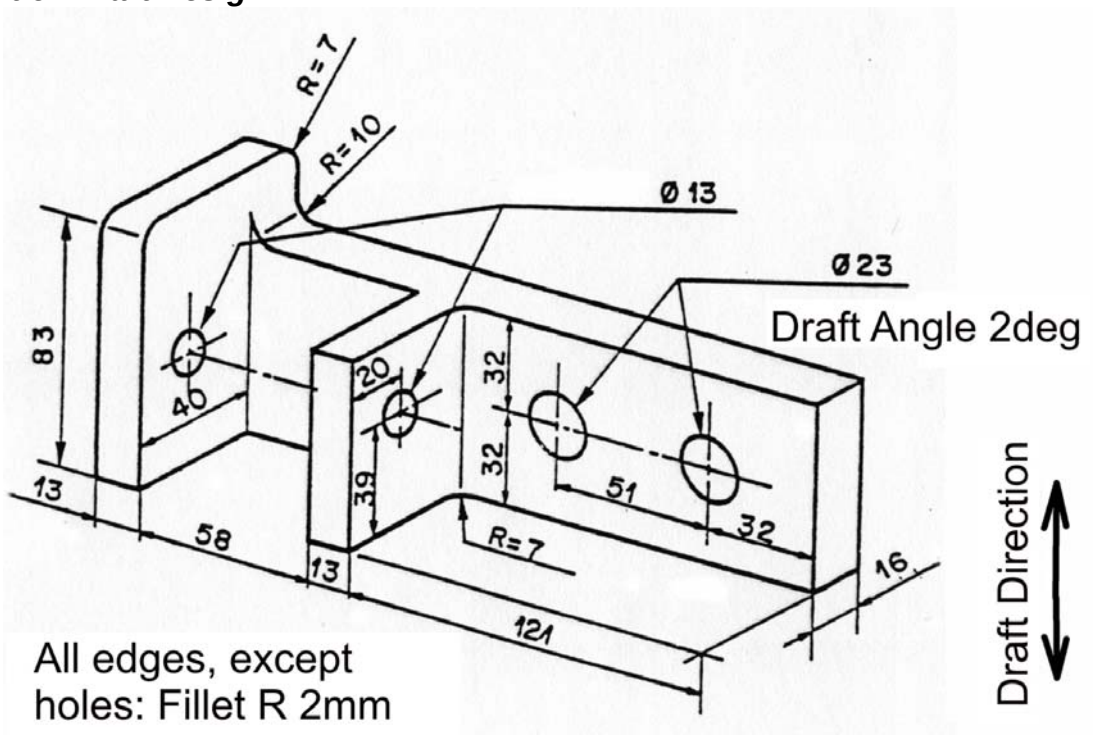


Figure 96: ***Angle anchor plate with holes***

Example 17: Machined part

Intention: Part Design

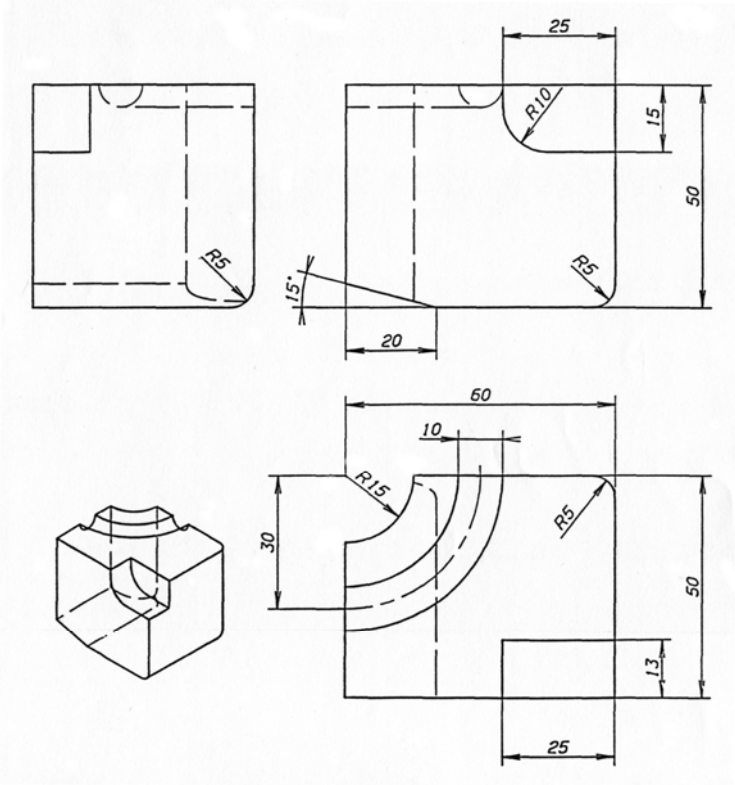
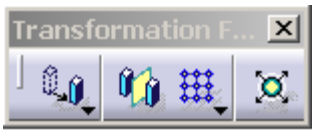


Figure 97: Body

6.4 Transformation Features



Transformation operations can be performed on bodies or parts of bodies, using the *Transformation Features*.

Translation, Rotation, Symmetry: Translates, rotates or mirrors (without duplicating) bodies. These features are able to transform the selected solids without displacing their basic geometry (sketches etc.). Otherwise, design elements can be positioned using the compass or 3D Constraints.

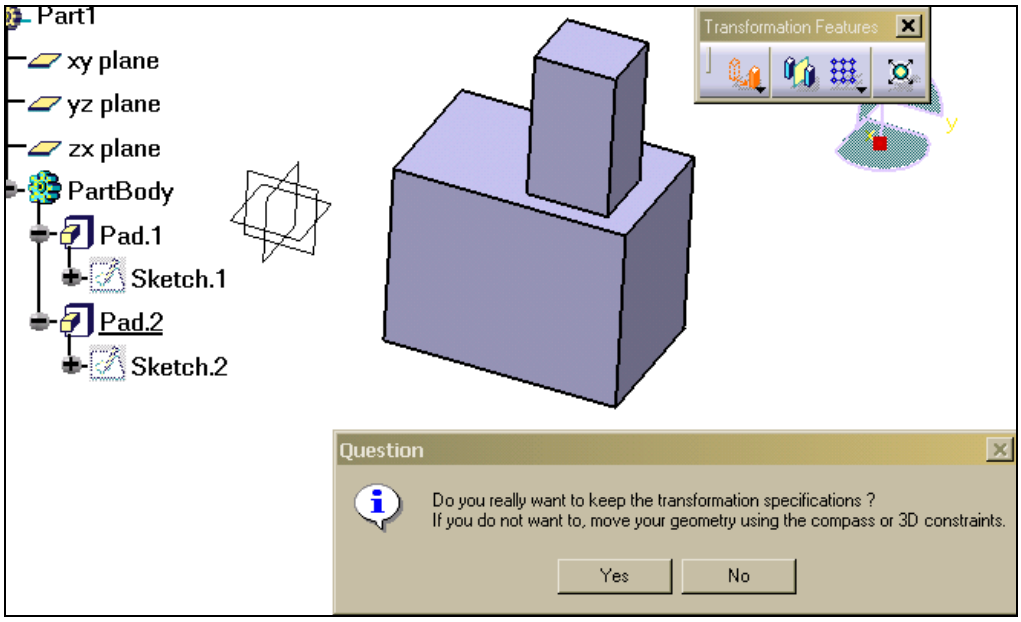


Figure 98: Starting a *Transformation Feature (Translation, Rotation* or *Symmetry)*

Mirror:

This symmetry function is duplicating the original mirrored element. It can be used to mirror either bodies or parts of bodies. Therefore, the geometry should be set active (or marked) in the specification tree BEFORE selecting the *Mirror* button.

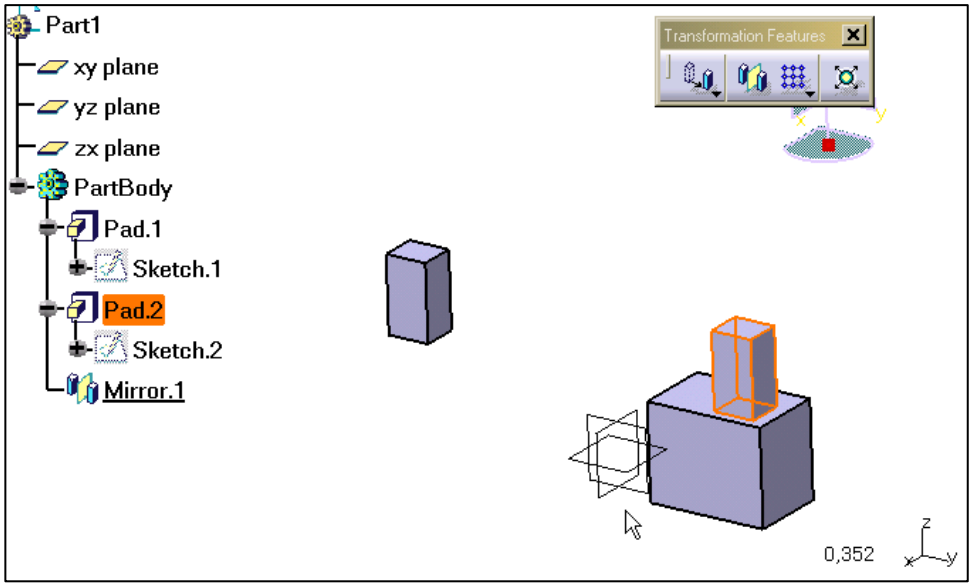


Figure 99: Mirroring a *Pad*

Patterns:

The *Pattern* features (*Rectangular Pattern, Circular Pattern* and *User Pattern)* enable the duplication of design elements using special positioning definitions.

The *Rectangular Pattern Definition* requires two translation directions, the according *Instance* number and the *Spacing. Parameters* offers several options for the input.
The *Reference Element* defines the translation direction, the *Object to Pattern* is the element that is duplicated.
The second direction is defined in a similar way.

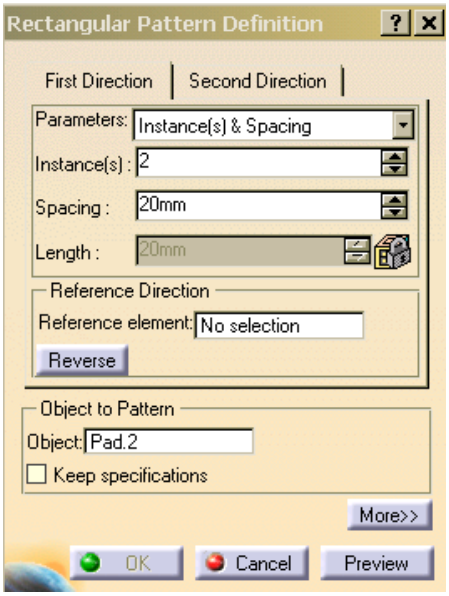


Figure 100: *Rectangular Pattern Definition*

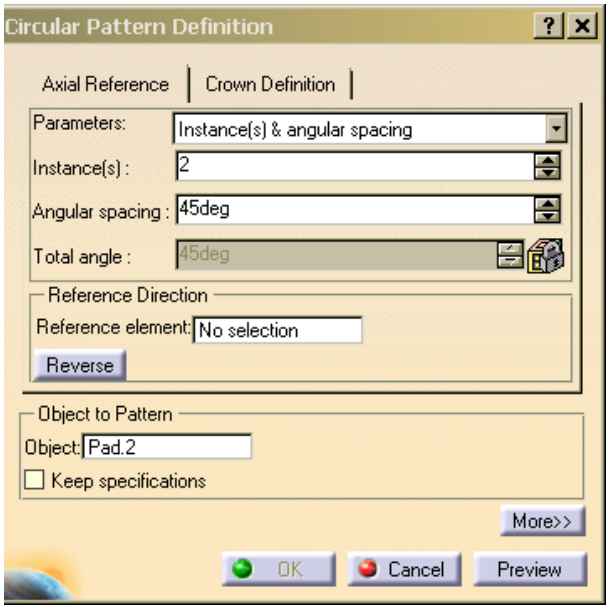


Figure 101: *Circular Pattern Definition*

The dialog box of the *Circular Pattern* requires the *Instance* number, the *Angular Spacing* and the definition of a *Reference Element* defining the rotation axis. *Crown Definition* offers more input options concerning a pattern in radial direction.

The feature *User Pattern Definition* supports the individual definition of the duplication of geometrical elements. To define the positions, a sketch can be created, containing anchor points.

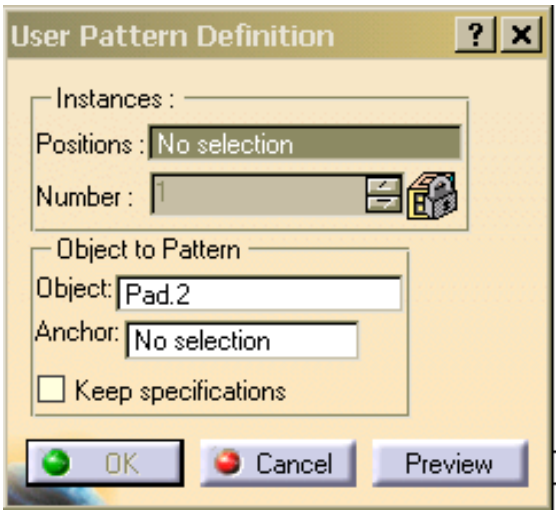


Figure 102: *User Pattern Definition*

Scaling:

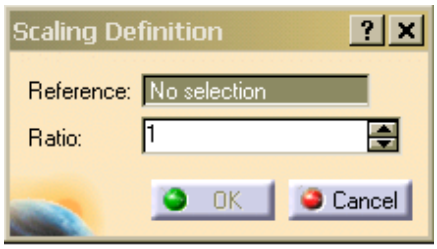


Figure 103: *Scaling Definition*

This feature scales elements to the specified dimension. Scaling can be carried out with respect to a point or a direction (line or normal to a plane).

Example 18: Drilled Panel

Intention: Transformation Features

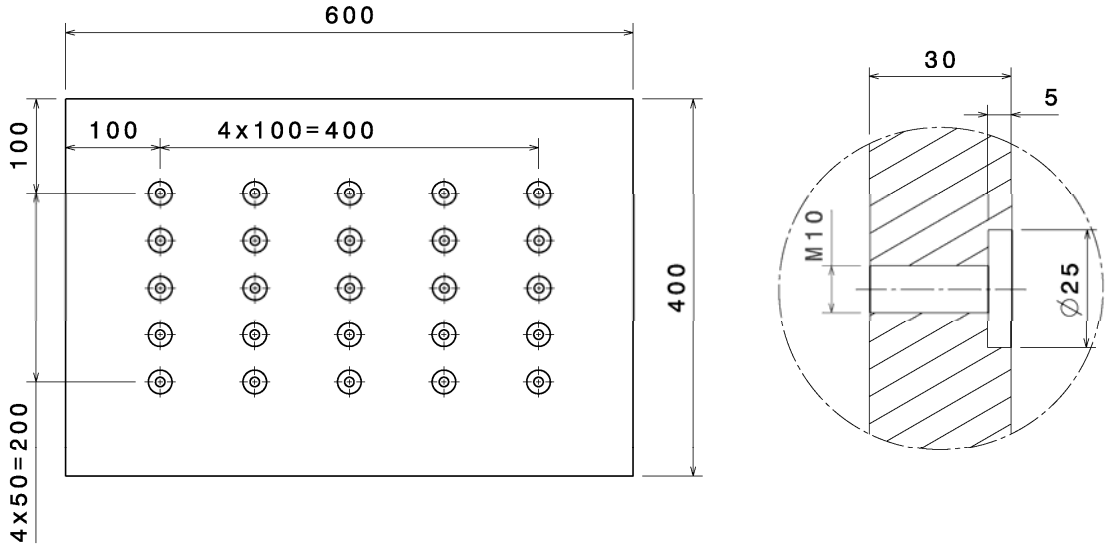


Figure 104: Drilled Panel

Beispiel 19: Angle bracket

Intention: Transformation Features

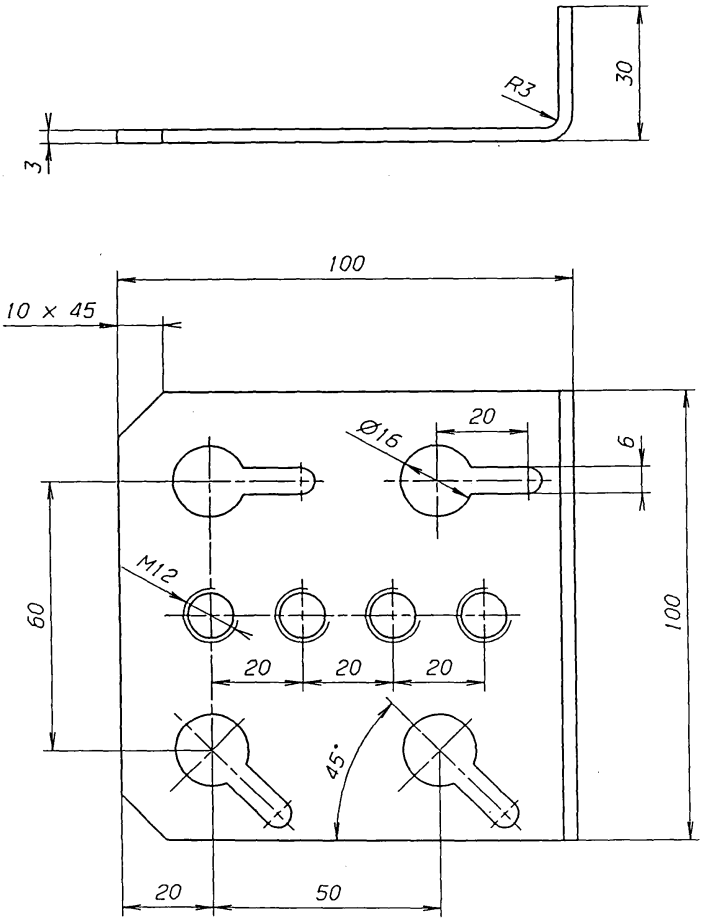


Figure 105: Angle bracket

Example 20: Asterisk shaped bracket

Intention: Transformation Features

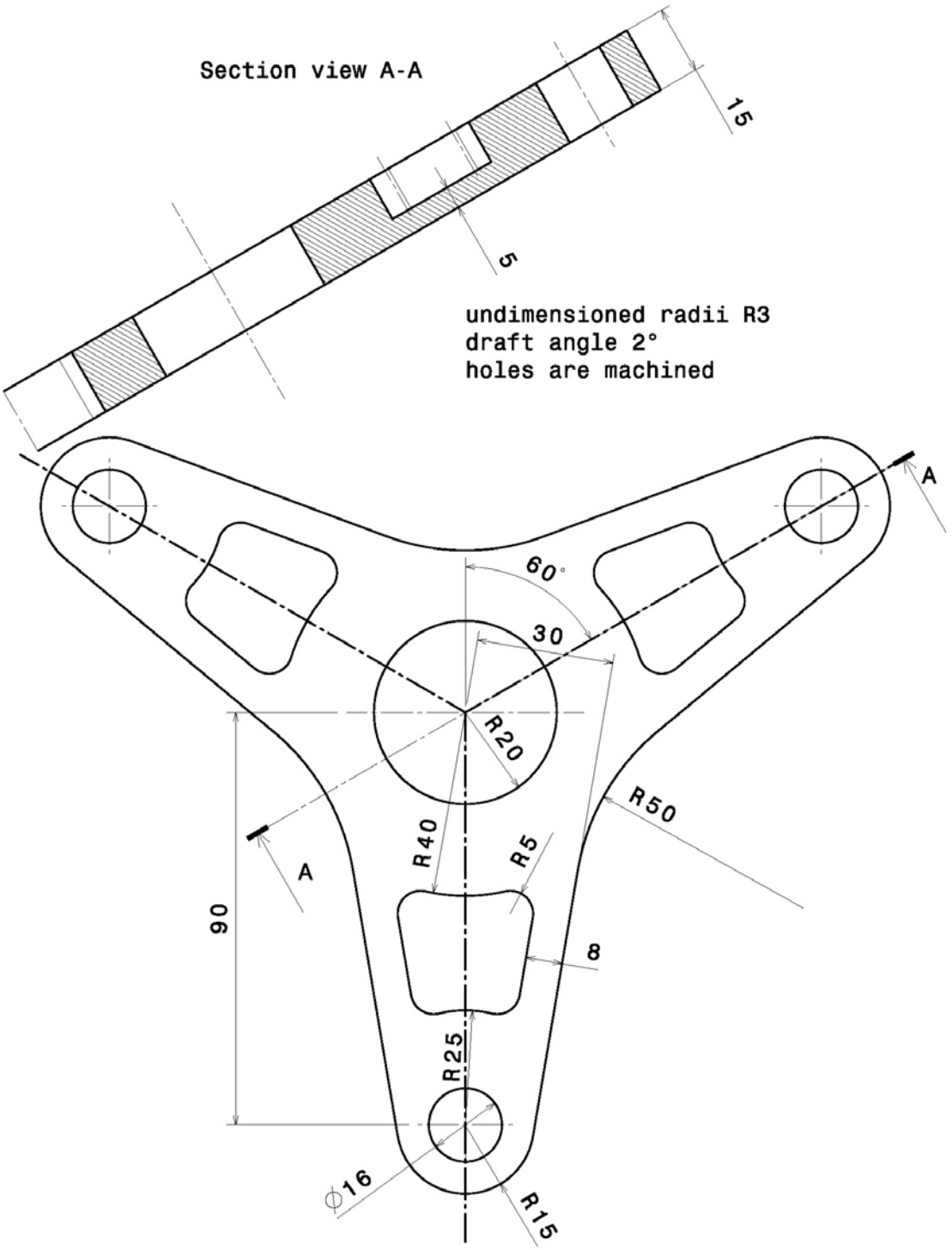


Figure 106: Asterisk shaped bracket

7 *Part Design* with several *Bodies* and *Boolean Operations*

The creation of complex bodies is supported by logic connection of *Bodies*. The logic operations (*Boolean Operations*) concern two bodies. The first body has to be active (*Define in Work Object*) and serves as a basic element. The second body affects the first one and is integrated in the first body in the *Specification Tree.*

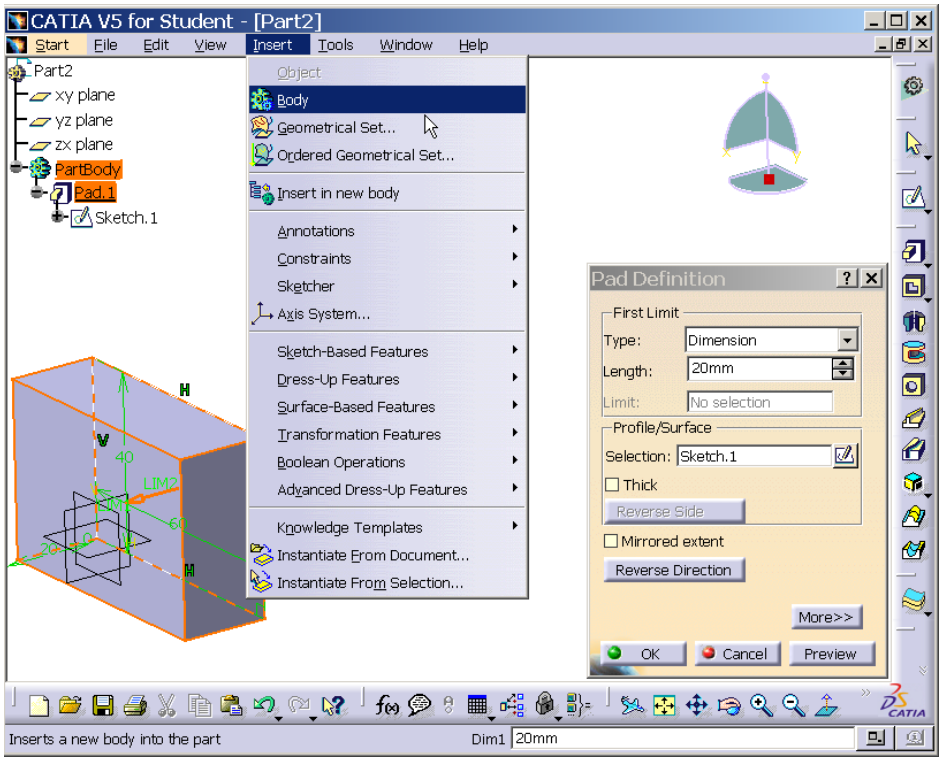


Figure 107: Inserting a new *Body*

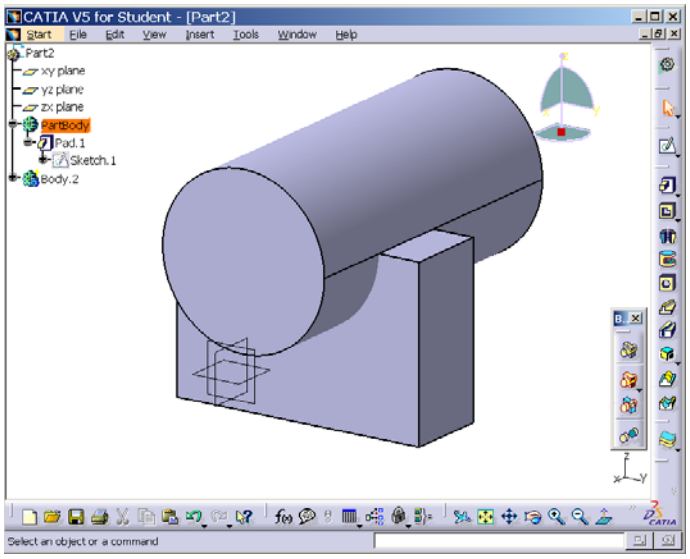


Figure 108: *Part* containing two *Bodies*

A new body can be inserted in the *Specification Tree* using *Insert / Body*. The new body is set the *In Work Object* automatically. The two *Bodies* are considered independent elements; one can be edited without affecting the other. The *Bodies* can be switched to the noshown space autonomously, for instance.

☺ **Annotation:** Modifications are carried out at the active body. Bodies can be activated by double clicking the desired branch of the specification tree (for features as *Sketch, Pad,* …), or by picking the element with the right mouse button and selecting *Define in Work Object* (for *Bodies*).

When clicking on an element with the right mouse button and picking *Properties*, user defined setting can be adjusted, such as renaming the design elements, changing the graphic representation or retrieving the object status.

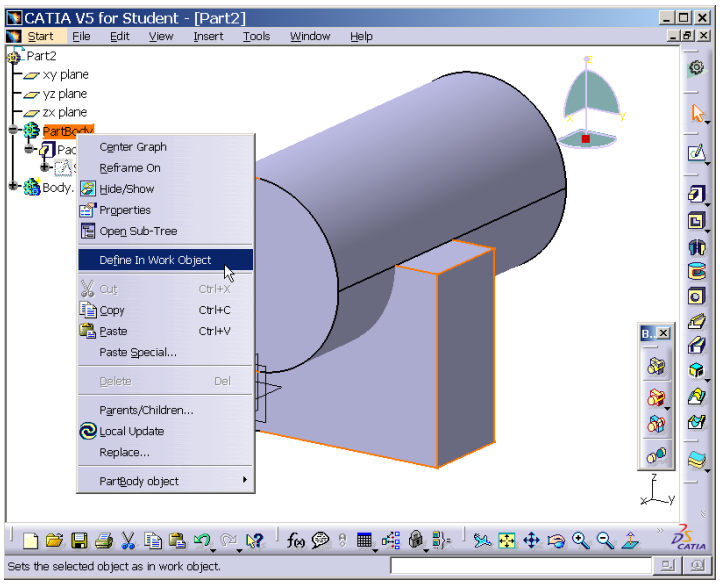


Figure 110: Defining the *In Work Object*

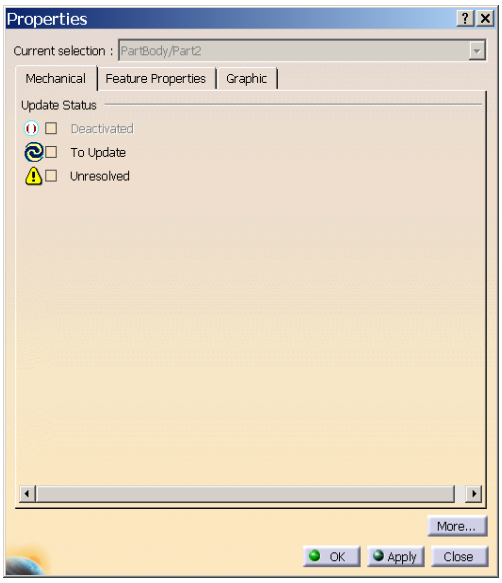


Figure 109: *Properties* dialog box

7.1 Boolean Operations

The *Boolean Operations* are accessed via the pull down menu *Insert / Boolean Operations* or the toolbar *Boolean Operations*.

The features can be divided into three groups:

- adding
- subtracting or
- intersecting

independent bodies.

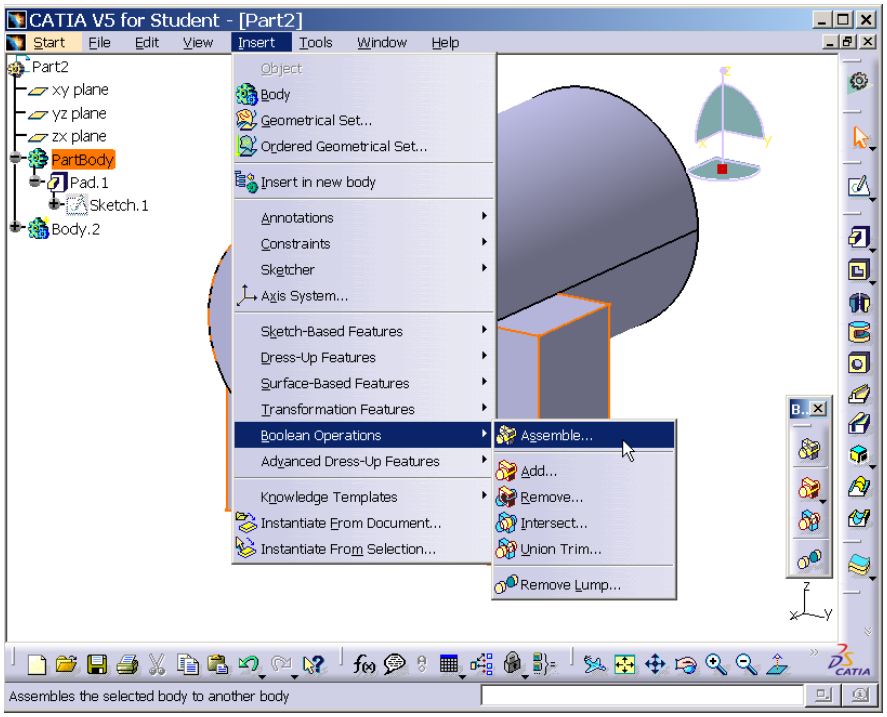


Figure 111: Accessing the *Boolean Operations* menu

This example uses two *Bodies*.
Figure 113 shows the result of a *Remove* operation; *Body 2* has been removed from *Part Body. Body 2* is integrated into the *Part Body* in the *Specification Tree*.

The following figures show miscellaneous *Boolean Operations*; all of them have been carried out using the same boundary conditions (*PartBody* as *In Work Object* and *Body 2* marked).

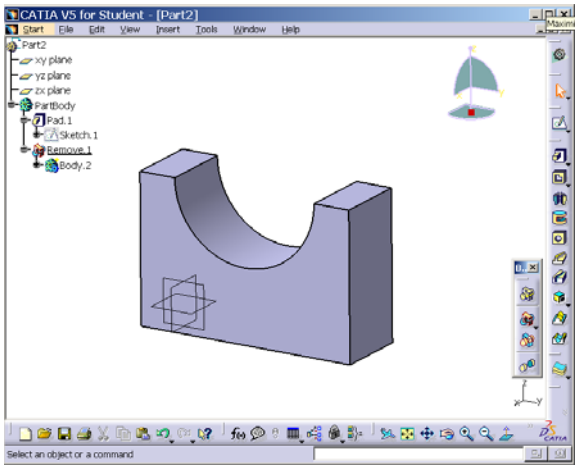


Figure 113: Result of *Removing Body 2* from *PartBody*

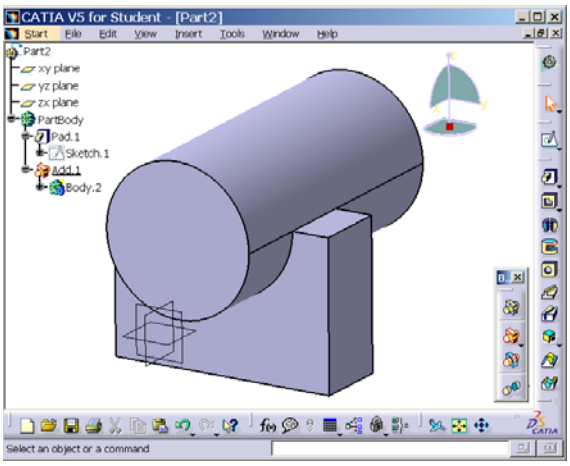


Figure 112: Result of *Adding PartBody* and *Body 2*

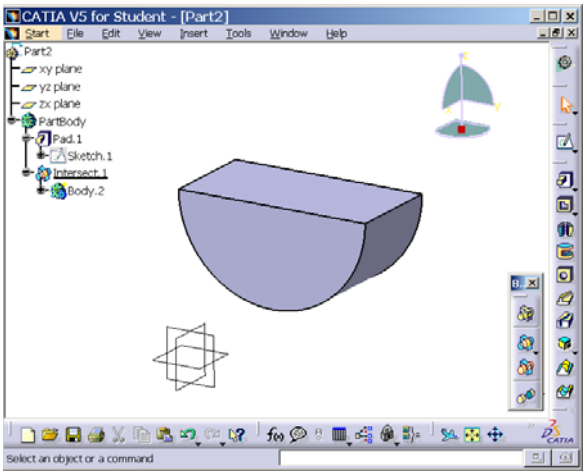


Figure 114: Result of *Intersecting PartBody* and *Body 2*

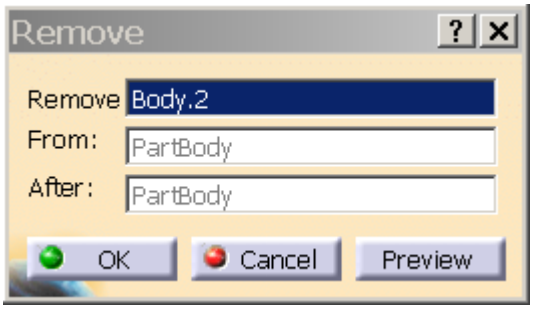


Figure 115: Dialog box of the *Remove* feature

The features *Assemble, Union Trim* and *Remove Lump* are exemplified in the following examples.

☺ **Annotation:** CATIA regards the *PartBody* as the ‚first' basic body. It can be modified using construction features, but the *PartBody* cannot be removed. Therefore it is important to consider the structure of the specification tree, especially for more complex parts.
The *PartBody* can be changed: Select the *Body* that should be defined the new *PartBody* with the right mouse button and select *Body.x / Change Part Body.*

The following example is going to show how to use *Boolean Operations*.

Example 21: Piston of a two-stroke engine

Intention: Part Design including several bodies and Boolean Operations

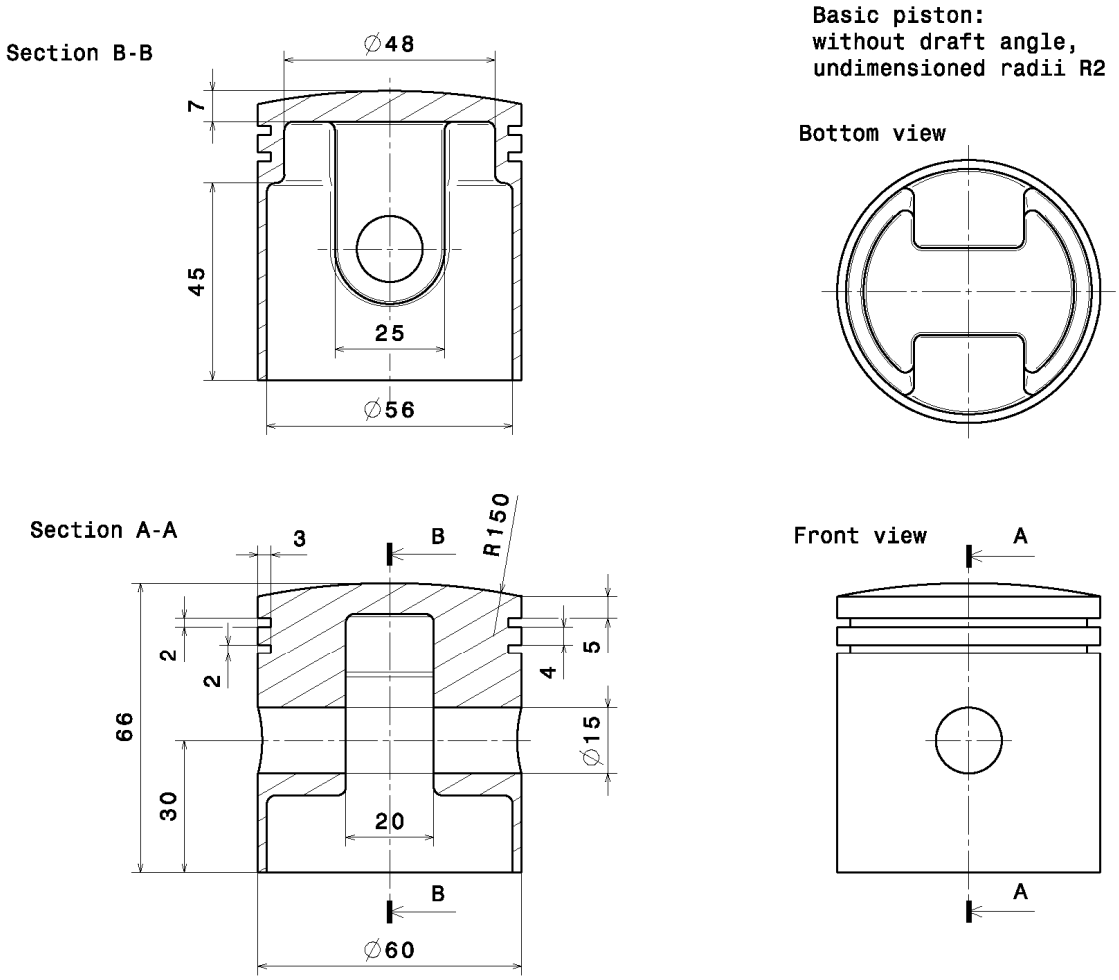


Figure 116: Piston of a two-stroke engine

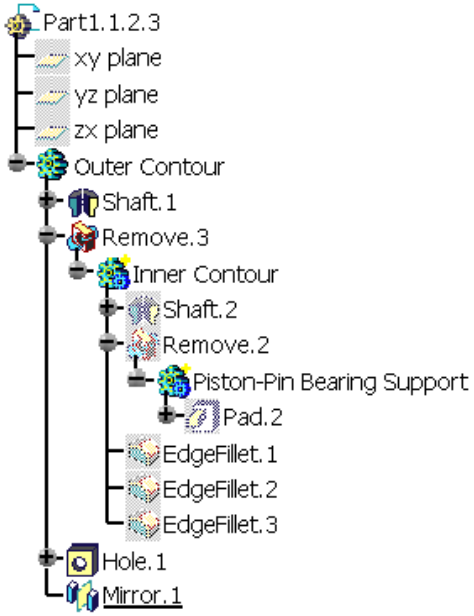


Figure 117: Specification tree of the piston

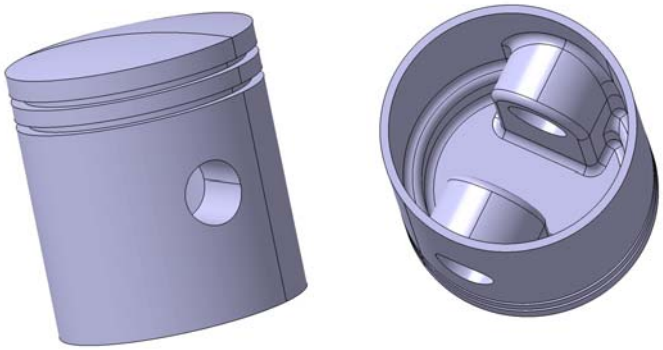


Figure 118: 3D views of the piston

A piston of a 175 ccm two stroke engine has to be designed. The two ring grooves are needed for the compression rings. The piston-pin bearing's diameter measures 15mm. For simplification reasons the grooves for the piston-pin retainers, the piston-ring retainers and the draft angles of the piston were disregarded. The dimensions of the piston are defined in Figure 116.

The *Specification Tree* shows that the part is built from three bodies:

Outer Contour
Inner Contour
Piston-Pin Bearing Support

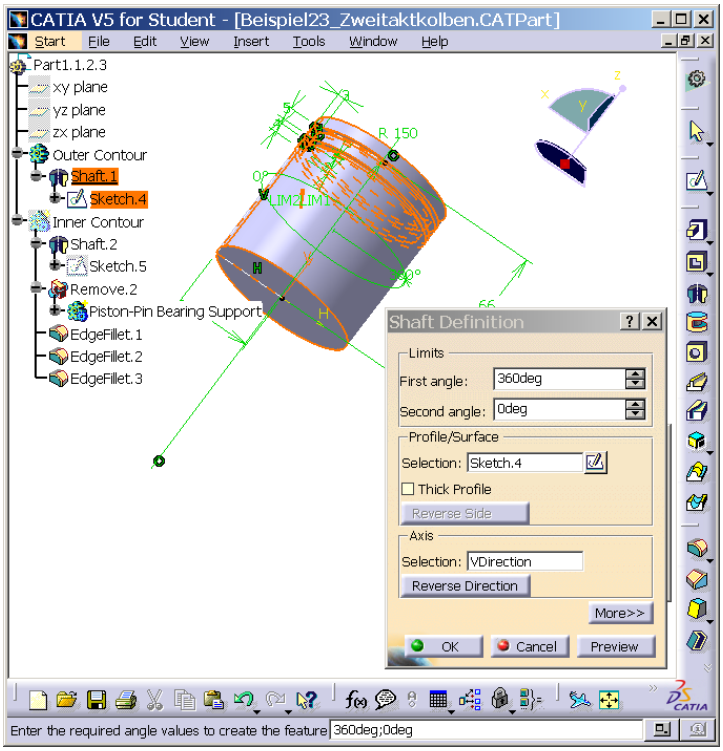


Figure 119: Creating the *Outer Contour*

The *Outer Contour* represents the outline of the piston. It can be created using the feature *Shaft*.

The inner contour consists of two bodies. The body *Inner Contour* consists of a *Shaft*, the *Piston-Pin Bearing Support* is a *Pad*.

The body *Piston-Pin Bearing Support* is being removed from the *Inner Contour* using the *Boolean Operation Remove*. It is commended that the *Fillets* be created as soon as possible, i.e. just after the *Boolean Operation*.

The *Inner Contour* is removed from the *Outer Contour*. At last the *Hole* for the *Piston Pin* is applied (feature *Hole*) and mirrored.

Of course other design strategies are possible. The example above shows a way that uses *Boolean Operations* to create a *Part* from several *Bodies*.

☺ **Annotation:** When designing a part that should be suitable for production, it is advantageous to design the outlines of the cast part and the machining separately. For the piston this strategy would result in an *Outer Contour* containing the draft angle and allowances. The machining is created in a body containing the final contour and the holes. One advantage of this strategy is the 3D model of the cast contour that can be directly accessed. The *Machining* body can be inactivated and the cast contour can be used as basis for the NC programming of the manufacturing machine for mould building.

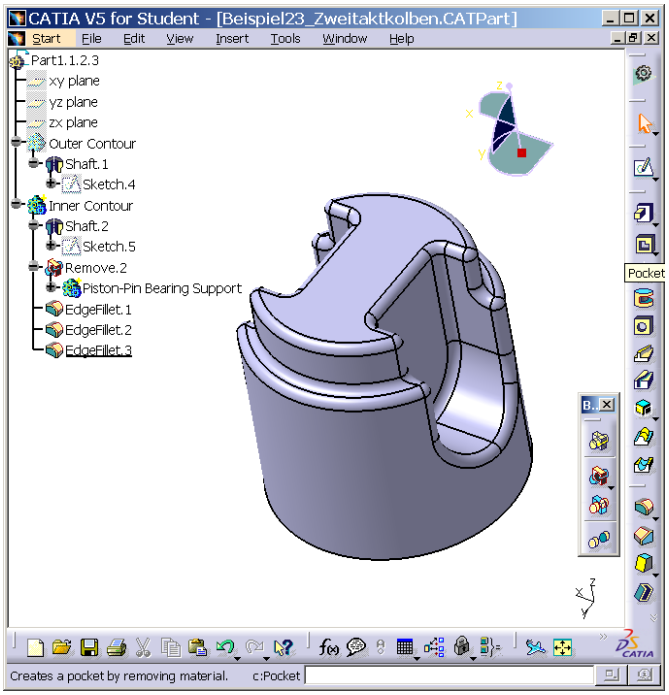
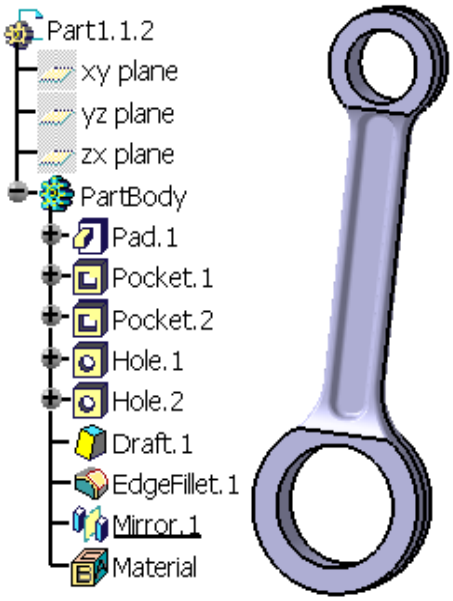


Figure 120: *Inner Contour* of the piston

Example 22: Conrod

Intention: Part Design



A conrod for a two-stroke engine has to be designed according to the drawing above. As displayed in the *Specification Tree*, the symmetry of the part can be considered. Special attention has to be paid to the fillets and the draft angles. (Annotation: The conrod is machined at the bearings and at the side contact faces of the bearings.)

Figure 121: Specification tree of the conrod

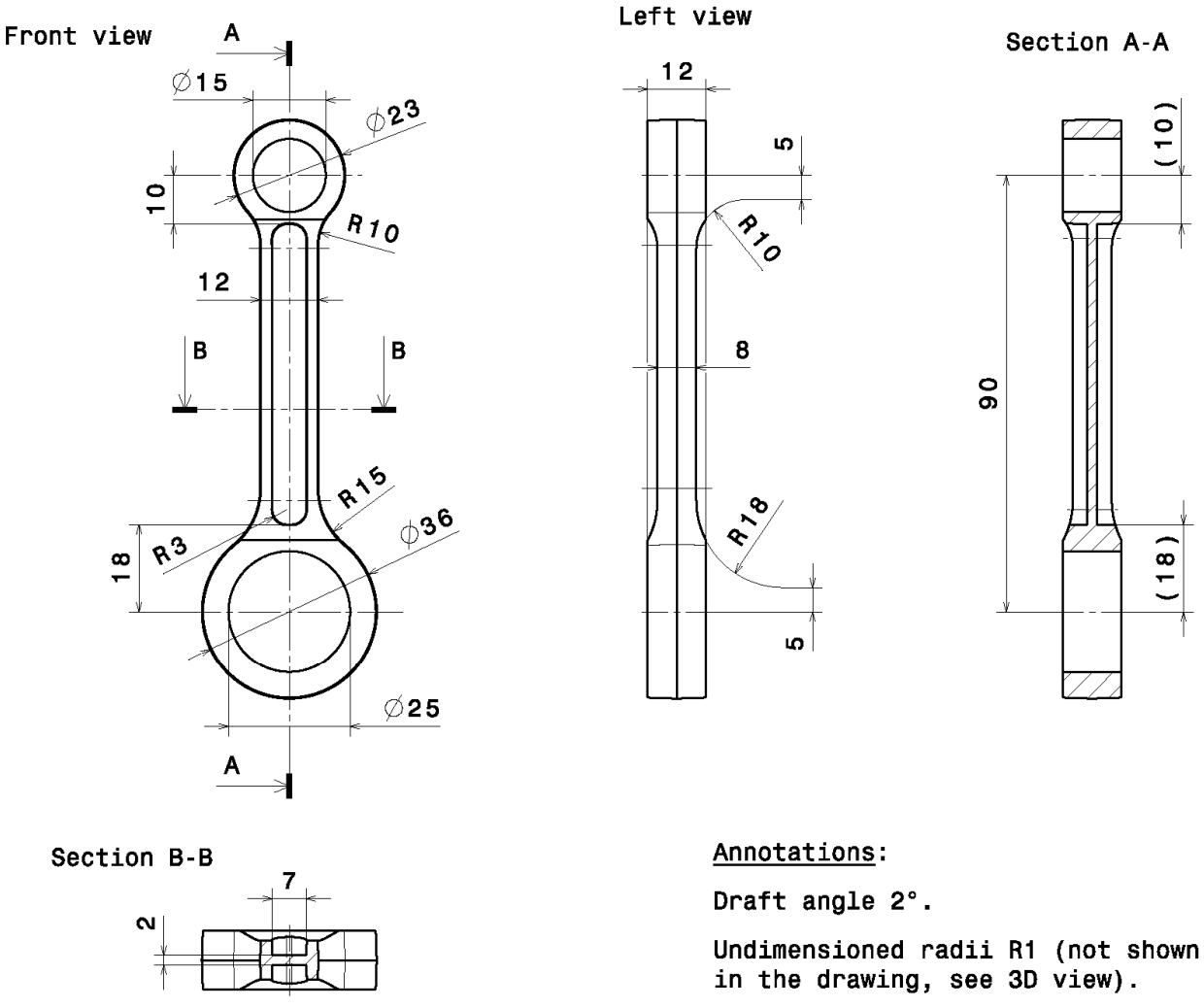


Figure 122: Conrod

8. The *Specification Tree* in *Part Design*

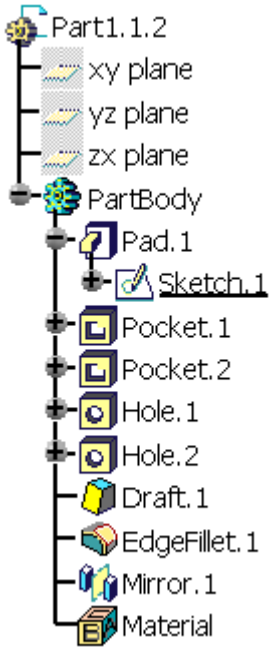


Figure 123: *Specification Tree*

The *Specification Tree* contains all elements of the actual *Part*. Not only the geometrical elements, but also the coordinate systems and constraints are shown. The levels are marked by vertical lines. The sub trees can be opened or closed in different ways, either by clicking on the nodes (+) or (-), or with the feature *Tree Expansion* in the pull down menu *View*.
When clicking on a branch of the *Specification Tree*, the tree can be edited and the *Part* itself is shaded dark. Clicking on a branch again reactivates the *Part*.
The active element of the *Part* is underlined in the *Specification Tree* (Figure 123: *Sketch.1*).To activate another element, click on it with the right mouse button and choose *Define In Work Object*.
To hide (or show) the *Specification Tree,* hit the F3-button.

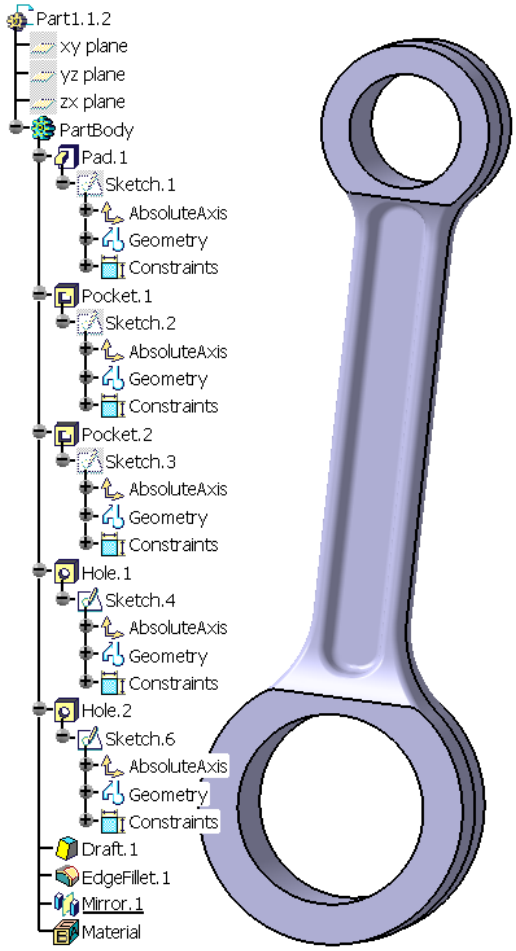


Figure 124: *Specification Tree* of the conrod

Figure 124 displays the *Specification Tree* of the conrod, showing a typical structure of a simple *Part* in CATIA V5. The conrod is built from a basic *Body* (*Pad.1*) that is modified using *Pockets* and *Holes*. The *Dress-up Features Draft* and *Edge Fillet* complete the part, which represents one half of the conrod. At last it is duplicated using the *Transformation Feature Mirror*.

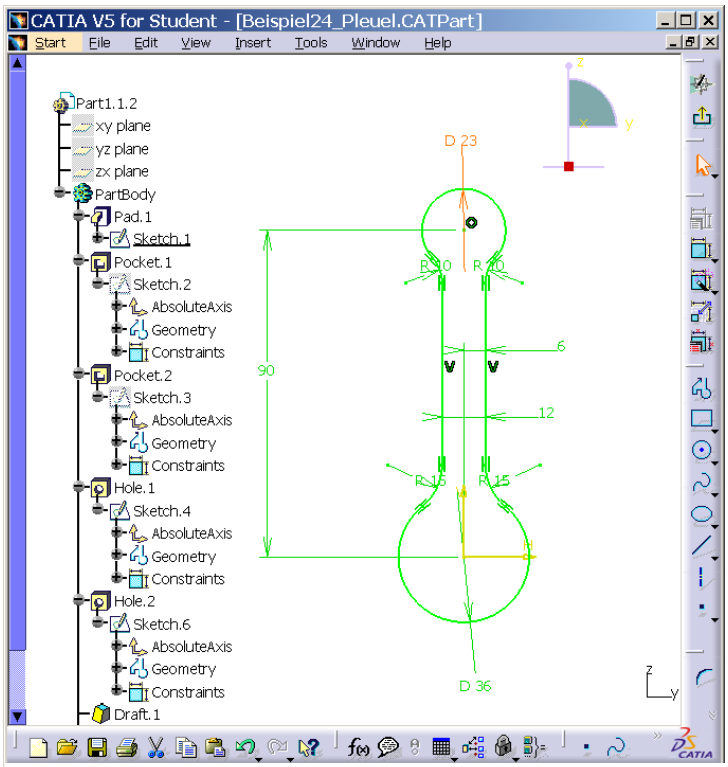


Figure 125: Modifying *Sketch.1*

The input parameters of the single elements can be modified by double clicking them. To change *Sketch.1* for example, double clicking it opens the sketch mode. The modifications of the sketch are applied to the element automatically after leaving the *Sketcher*.

Similarly, the parameters of all features and operations can be changed. Figure 126 shows the dialog box that appears after double clicking *Pocket.1* in the *Specification Tree*. The according feature is activated (→ underlined in the tree) and the definition window opens. CATIA changes the display mode of the body and shows the modifiable parameters.

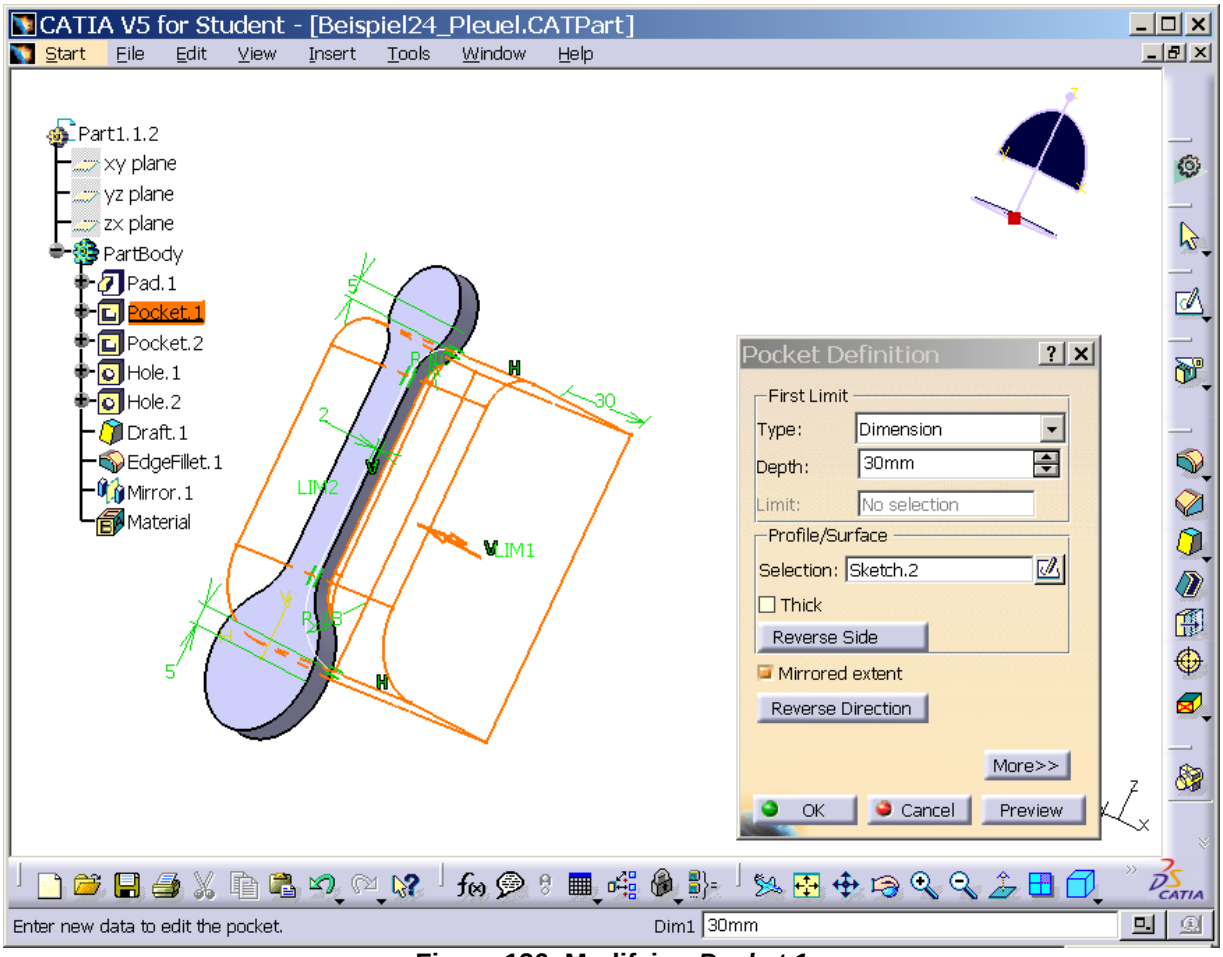


Figure 126: Modifying *Pocket.1*

A *Part* consisting of several *Bodies*, connected e.g. via *Boolean Operations*, can be modified by accessing the *Specification Tree* that offers a well structured overview of the build-up. Figure 127 shows the *Specification Tree* of a piston. The part is structured in a basic *Body* (*Outer Contour*) and a removed *Body* (*Inner Contour*). The two *Bodies* are connected with the *Boolean Operation Remove.3*.

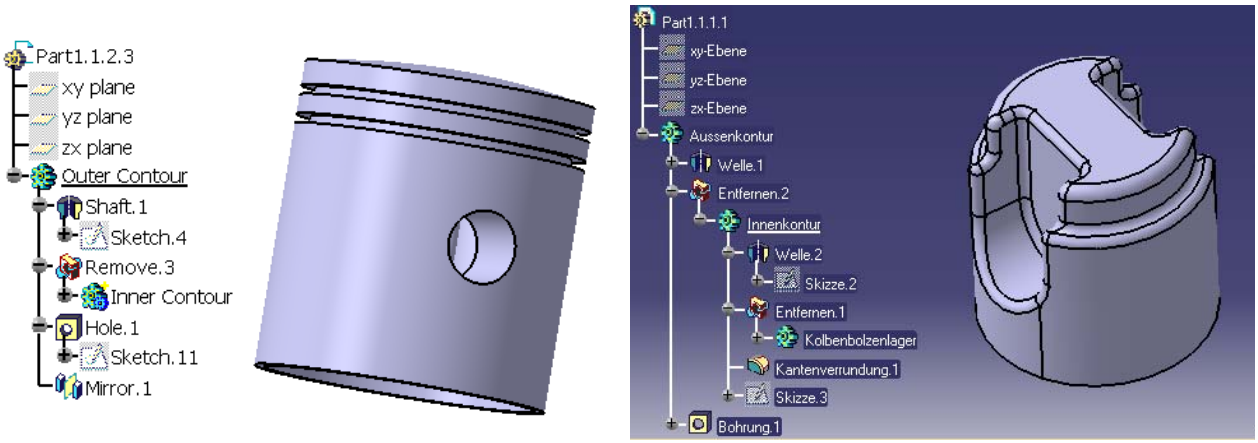


Figure 127: Piston

After activating the removed *Body* (*Inner Contour*), only the elements of the active branch of the tree are displayed. The buildup of the *Inner Contour* (it consists of two *Bodies* as well) can be seen. Activating the *Part* step by step in the *Specification Tree* offers a helpful overview of the construction and enables quick comprehension of the structure and easy modification of parameters when editing the part. CATIA V4 used the feature *Smart Solid* to analyse the *Specification Tree* and the *Part* structure. In CATIA V5, this feature has been integrated in the *Specification Tree*, offering the possibility to set single elements *In Work*.

☺ **Annotation:** The display mode of elements that are activated in the tree can be set by use of the button *Only Current Body* in the *Tools* toolbar. If this option is activated, only the active body is displayed; otherwise all bodies are visible. This option can also be set in the pull down menu *Tools* / *Options* / *Infrastructure* / *Part Infrastructure* / *Display* / *Display in Geometry Area* / *Only the current operated Solid* or *Only the current Body.*

☺ **Annotation:** Specific component adjustments can be set by clicking on the *Specification Tree* with the right mouse button. The *Properties* menu allows several options, such as switching elements from / to the noshown space (*Hide / Show*), rename an element or change its appearance. (See chapter *5.3 Structure of the specification tree of a sketch.*)

☺ **Annotation:** Objects can be moved within the *Specification Tree* by picking them with the right mouse button and selecting *Object X* / *Reorder*. The dependencies of the moved objects have to be considered.

Example 23: Prism body

Intention: Part Design

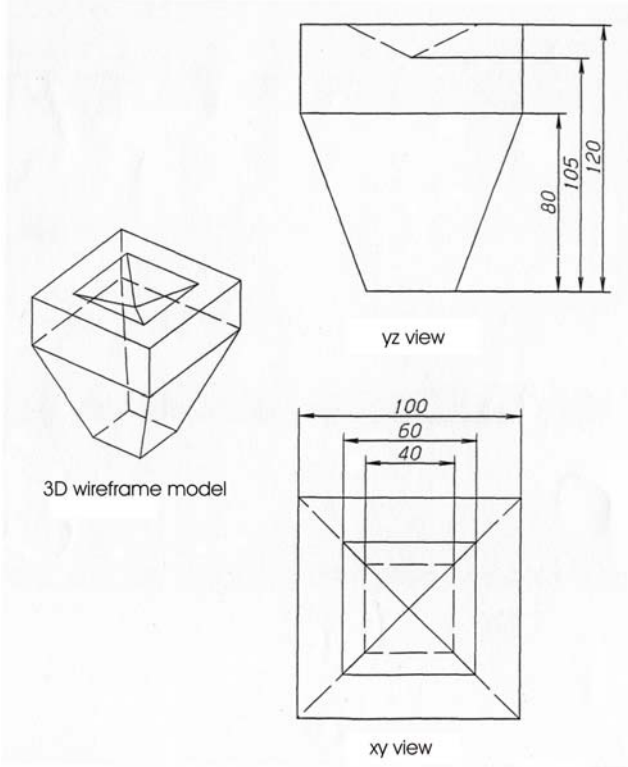


Figure 128: Prism body

Example 24: Pendulum

Intention: Part Design

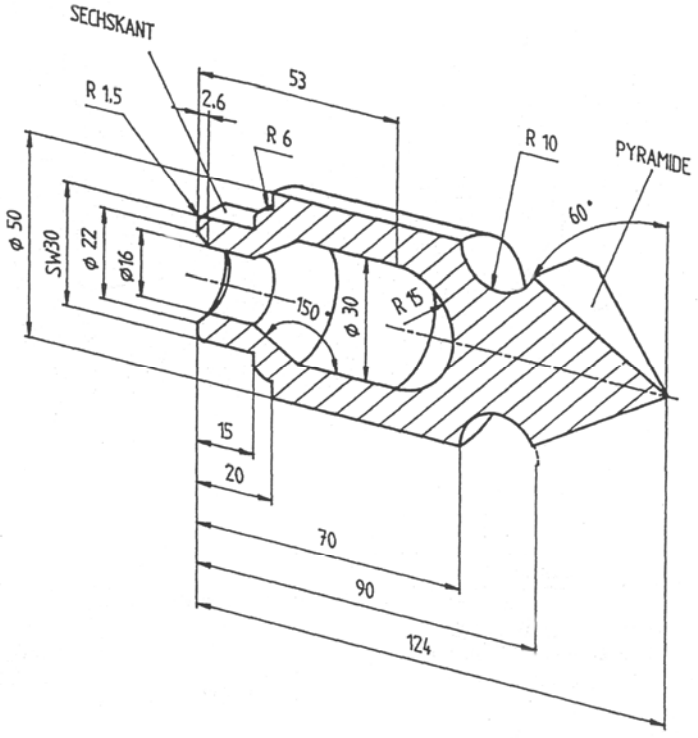


Figure 129: Pendulum

Example 25: Adjusting wheel

Intention: Part Design

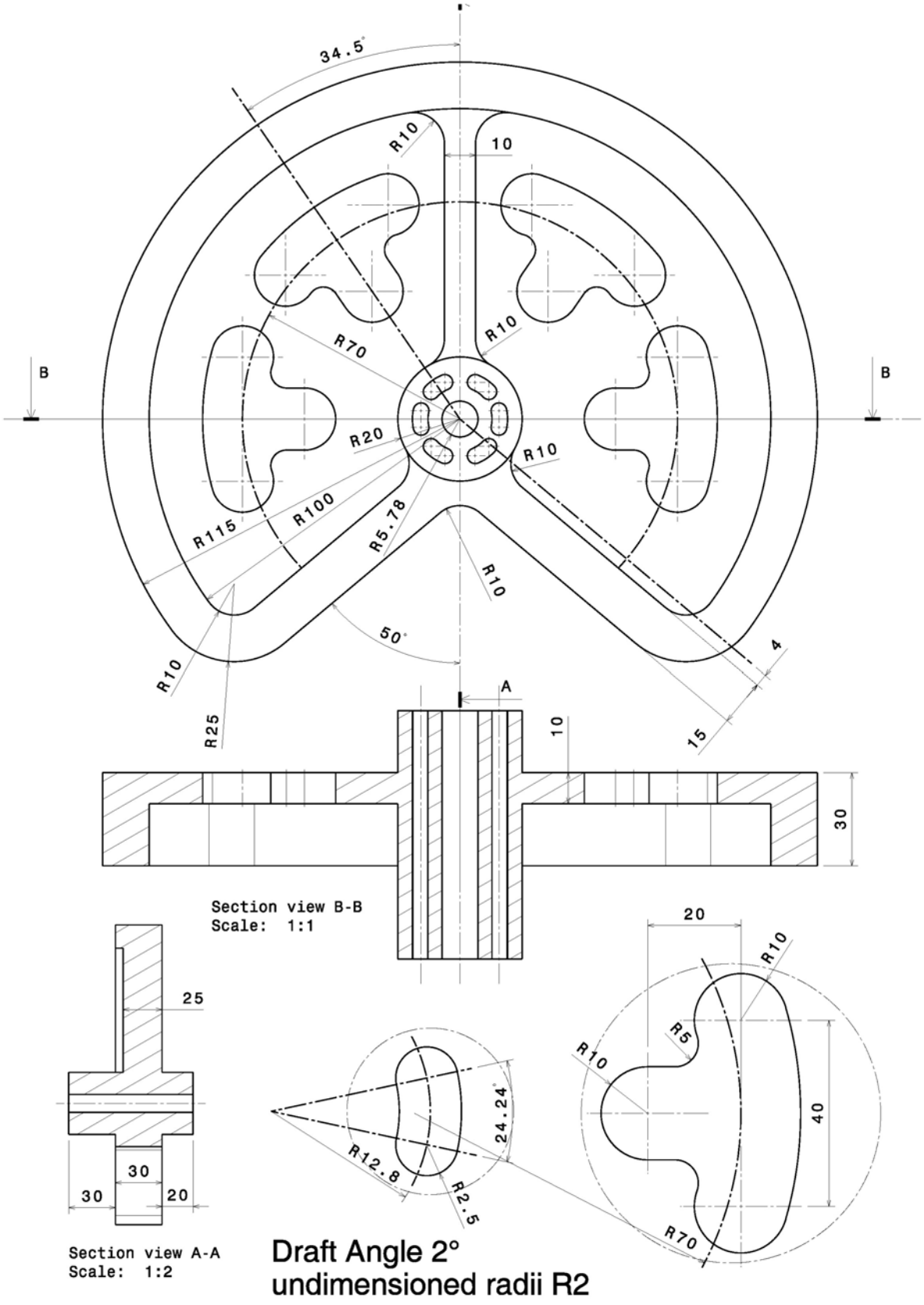


Figure 130: Adjusting wheel

9 Creating assemblies in the workbench *Assembly Design*

The workbench *Assembly Design* is used to integrate several construction elements (*Parts* or *Products*) into one *Product*. Thus, a *Product* specifies the assembly of components that are positioned according to user requested constraints. The workbench is activated by calling up a new *Product* in the *List of Types* window (pull down menu *File / New*).

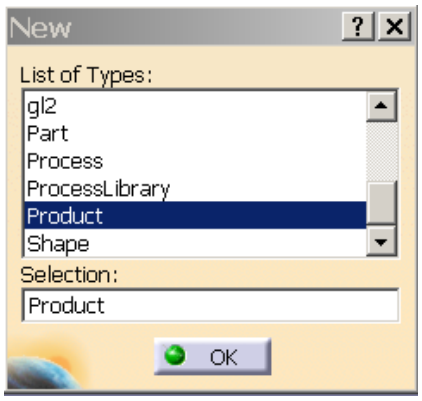


Figure 132: Creating a new *Product*

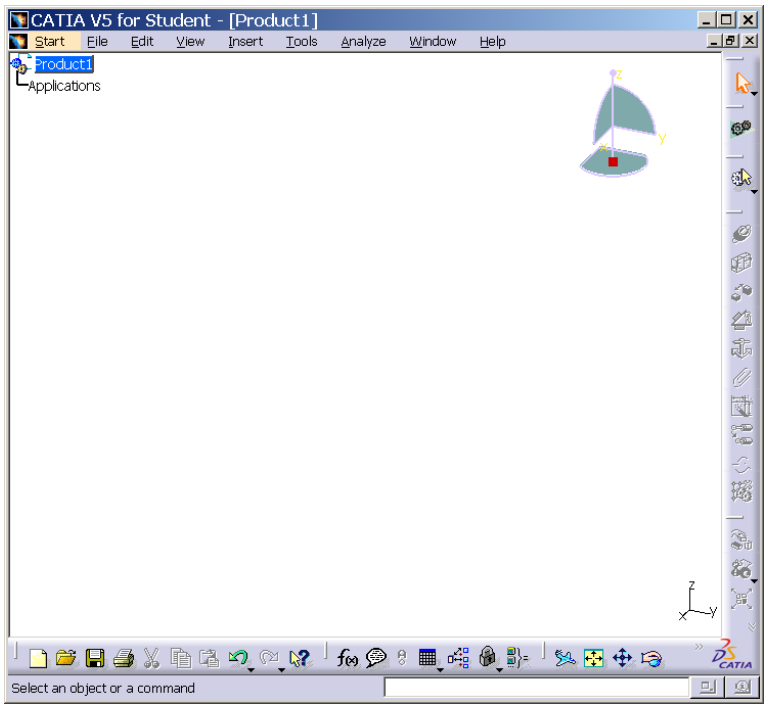

CATIA V5 for Student - [Product1]
Start File Edit View Insert Tools Analyze Window Help
Product1
Applications
Select an object or a command

Figure 131: The *Assembly Design* workbench

This chapter treats the most frequently used basic features of *Assembly Design*. They are subsequently exemplified in a simple example. The *Product* generation represents the basis for further CATIA V5 tools, such as the features in the DMU (*Digital Mockup*) workbench.

9.1. Operations in the Assembly Design mode

9.1.1 Product Structure Tools

Component ... Inserts a new *Component* in the selected *Product*
Product ... Inserts a new *Product* in the selected one
Part ... Inserts a new *Part* in the selected component
Existing Component ... Inserts an existing component in the selected one
Existing Component With Positioning ... Inserts an existing component in the selected one and launches a command to position it
Replace Component ... Replaces a component by another existing one
Graph Tree Reordering ... Changes the order in the *Graph Tree* of the children of the selected component
Generate Numbering ... Generate a number on all components owning a representation
Selective Load ... Select instances of the *Products* which must be loaded

Manage Representations ...	Manages the representations of the selected component
Fast Multi Instantiation ...	Define and create multi instantiations

9.1.2 Constraints

The four constraints *Coincidence, Contact, Offset* and *Angle* are used to position components to each other. To fix the *Product* in space, one component can be fixed using *Fix Component*.

Coincidence Constraint …	Create a coincidence constraint between two components of the active component
Contact Constraint …	Create a contact constraint between two components of the active component
Offset Constraint …	Create an offset constraint between two components of the active component
Angle Constraint …	Create a coincidence constraint between two components of the active component
Fix Component ...	Fix the component position in the active component
Fix together ...	Create a relation between the selected components which fix their relative location
Quick Constraint ...	Automatically constraints the selected components
Flexible / Rigid Subassembly ...	Allows to overload position of child components of the product instance
Change constraint ...	Changes the type of the selected constraint
Reuse pattern...	Reuse pattern feature of a part to instantiate a component

9.1.3 Move

The toolbar *Move* is used to position single components of a product in space without using *Constraints*. The dialog box of the feature *Manipulation* shows possible ways of moving and rotating components along / around the x, y and z axis or user defined axes. If some constraints have already been defined, they can be taken into account using *With respect to constraints*.

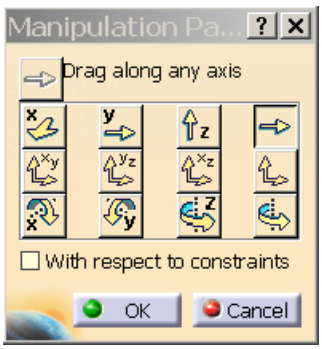


Figure 133: Dialog box of the *Manipulation* feature

Manipulation ...	Move a component by freehand translation or rotation with the mouse
Snap ...	Move a component onto another component by snapping
Explode ...	Creates a 3D exploded view
Stop manipulate on clash ...	Stop manipulation when a clash occurs

9.1.4 Space Analysis

The features *Clash, Sectioning* and *Distance and Band Analysis* enable a check of components in a *Product* concerning their relative position.

The dialog box *Clash* offers options for the *type* of interference (*Contact+Clash, Clearance+Contact+Clash, Authorized Penetration, Clash Rule*) and the affected components.

Figure 134: *Clash dialog* box

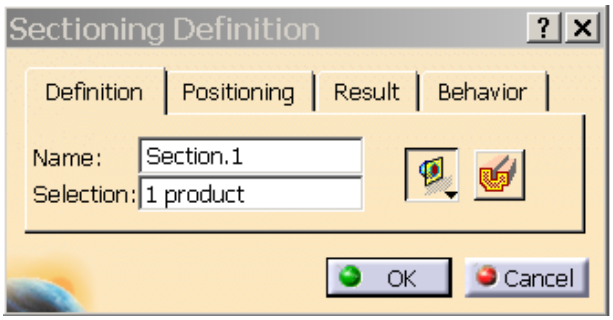


Figure 135: *Sectioning Defintion* window

The feature *Sectioning* is available for quickly creating sections of a *Product*. A plane is needed to define the section, which is displayed in an additional window. The section can be exported in different file types (e.g. dxf, igs, etc.). Additionally, the definition can be modified later on.

The feature *Distance and Band Analysis* can be used to measure distances between components. The definition box enables the input of the *Accuracy* and the desired distance orientation.

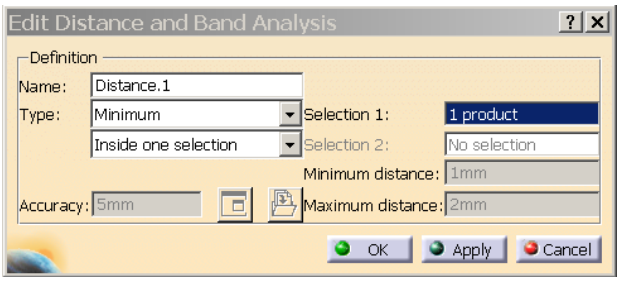


Figure 136: *Distance and Band Analysis* definition

9.1.5 Update

The set constraints are realized in the *Product* after *Updating* the *Product* or parts of it.
Parts that have been manipulated manually without consideration of constraints are reset to the position defined by the constraints when updated. After having modified the specification tree, the *Product* has to be updated as well.

☺ **Annotation:** Similar to other design modes (*Part Design, Drafting*), the properties menu, activated by right-clicking the *Specification Tree,* contains helpful features such as *Hide / Show* and other component specific definitions.

9.2. The Specification Tree in Assembly Design

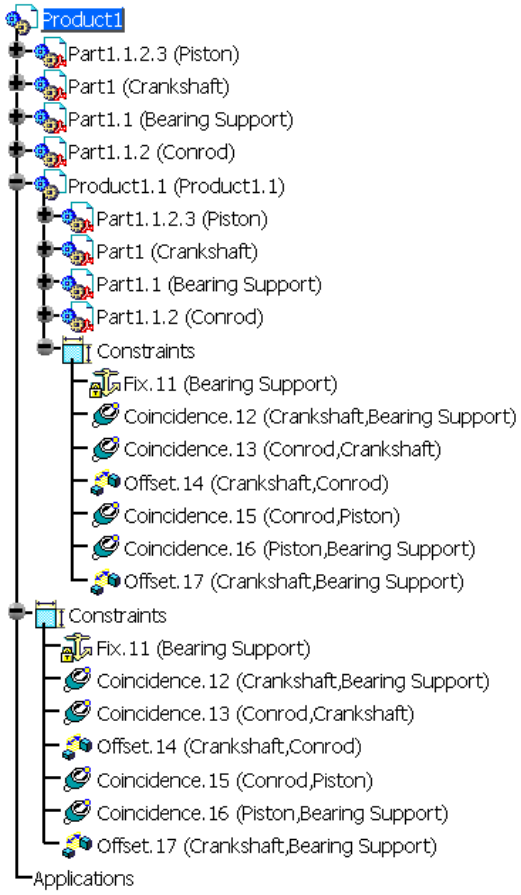


Figure 137: *Specification Tree* of a *Product*

In a *Product Specification Tree* the components of the assembly (which can be mainly *Parts* or *Products*) are displayed according to their order. The actually active branch is marked blue.
Additionally, the defined *Constraints* are shown in the according layer in the tree. In case *Parameters* (*Formulas* or *Design Table*) have been used, they are shown in the tree as well as *Applications,* such as *Kinematics*, *Generative Structural Analysis* or other modules.
Via the *Specification Tree* it is possible to change from *Assembly Design* directly to *Part Design*. Therefore the respective *Part* has to be set *In Work*. The according branch in the tree is marked blue and the menu environment of *Part Design* is shown. By activating the whole *Product*, it is possible to get back to the *Assembly Design* workbench again. This method of quick switches between the single workbenches makes designing in *Products* simple and effective, because it enables easy modifications of *Parts*. Nevertheless it is necessary for the designer to consider the influences of modifications of *Parts* on the *Product*. Otherwise problems can occur concerning the positioning or definition of construction elements, especially, if *Formulas* are used to connect single elements.

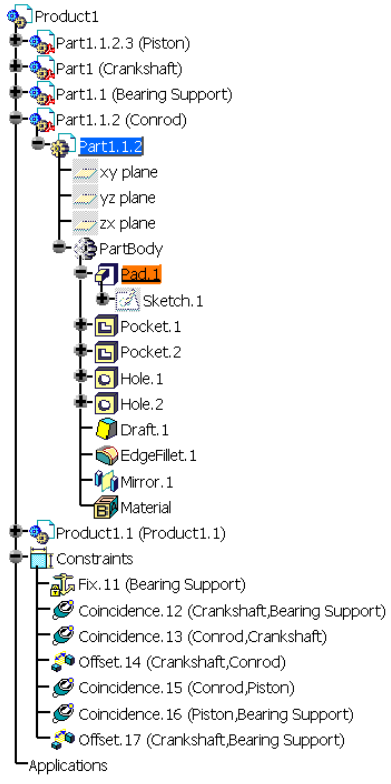


Figure 138: Editing a *Part*

The properties window (right mouse button; similar to the one in *Part Design*) contains some helpful design and definition features, such as defining *In Work Objects*.
With the properties window of a *Product*, a new component can be inserted in the assembly using the feature *Components*.

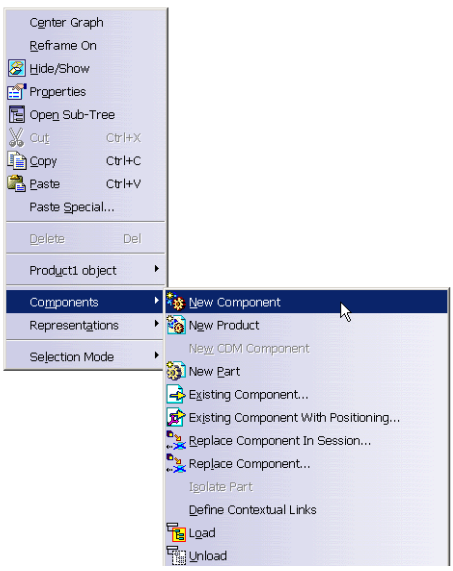


Figure 139: Inserting a new component

9.3. The Desk in CATIA V5

The *Desk* shows all links of an assembly, as there are links to *Components, Design Tables* or external objects that have been inserted. If there have been deduced *Drawings*, these links are displayed as well (if the *Drawings* are actually open).

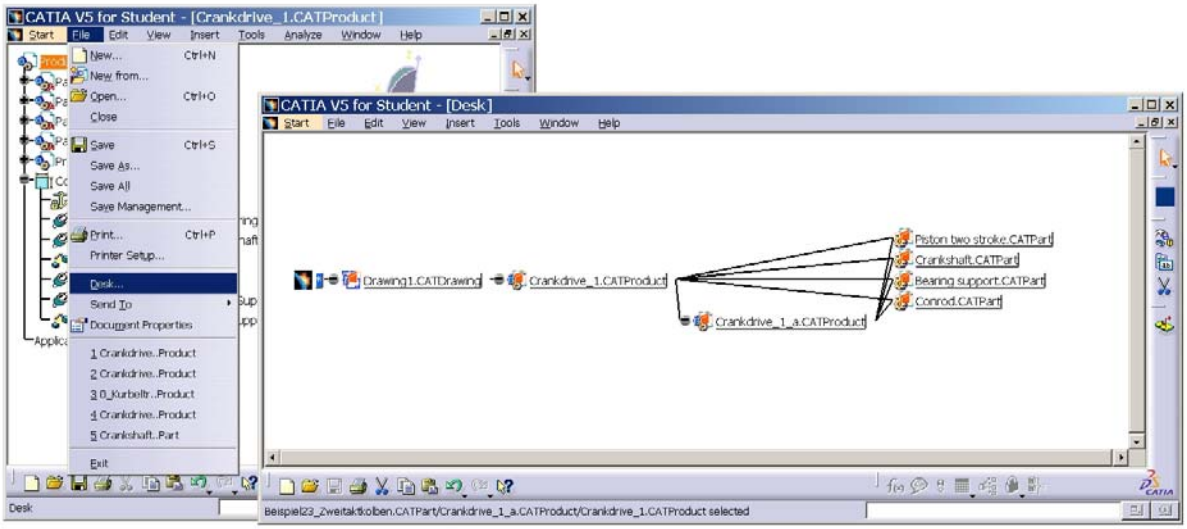


Figure 140: The *Desk* window

The pull down menu *File* is used to activate the *Desk*.
The user interface of the *Desk* (Figure 140) shows the link structure of the *Specification Tree* on the left side. The assembly consists of 2D *Drawing* (*Drawing1.CATDrawing*) and a *Product* (*Crankdrive_1.CATDrawing*); which itself consits of the components *Piston two stroke, Crankshaft, Bearing support, Conrod* and a subassembly (*Crankdrive_1_a*). The subassembly *Crankdrive_1_a* accesses, like the *Product Crankdrive_1*, the four *Components*.

By pressing the right mouse button, the property window of the component link can be opened (feature *Links*). This feature enables a remove-ment, creation or refreshment of links.

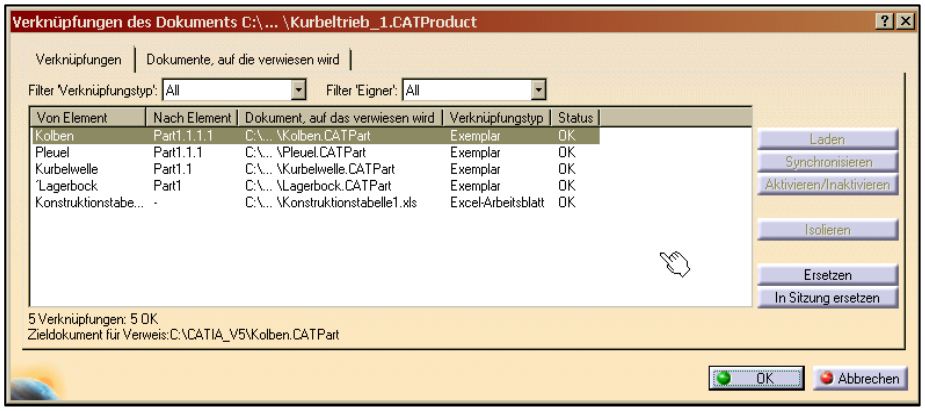
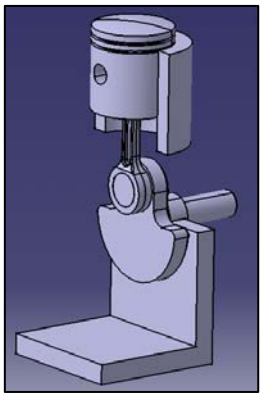


Figure 141: The *Links* window

☺ Annotation:
If cross linked files of CATIA assemblies are copied into other folders, the links have to be reassigned. This process is done in the *Desk*. CATIA automatically recognizes the components only in case the file *x.CADProduct* is in the folder of the linked components.

Example 25: Crank drive

Intention: Create an assembly



An assembly has to be created from the following components:

- *Piston* (*Piston of a two stroke engine*, Example 21)
- *Conrod* (Example 22)
- *Crankshaft*
- *Bearing support*

The *Crankshaft* and the *Bearing support* have to be designed according to the two *Drawings*:

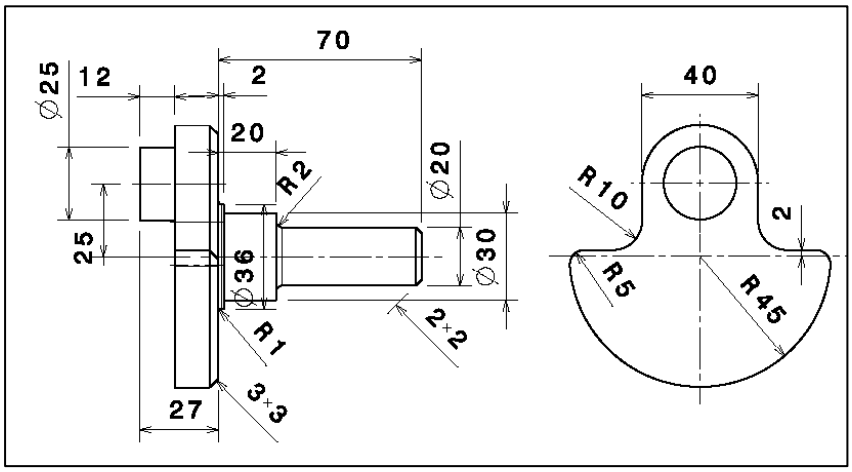


Figure 143: *Crankshaft*

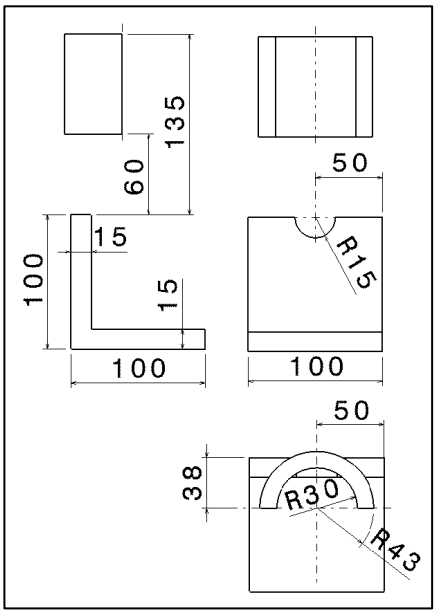


Figure 142: *Bearing support*

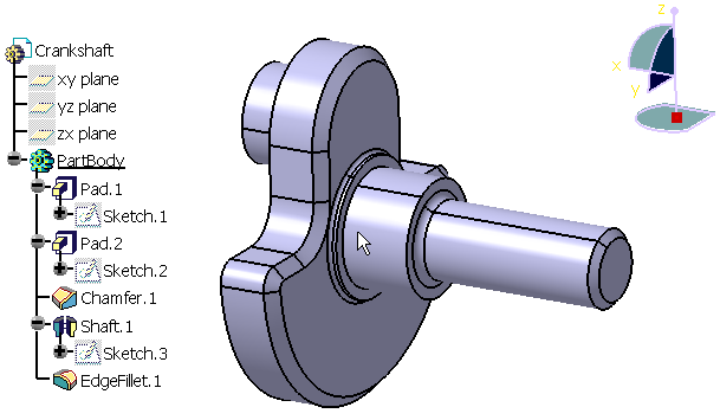


Figure 145: *Crankshaft*

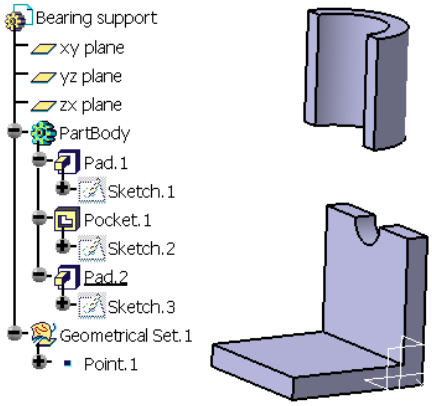


Figure 144: *Bearing Support*

After opening a new assembly (*File / New / Product*), the four existing *Parts* can be inserted using *Existing Component*. This feature can be activated either by the properties menu (right mouse button) *Components*, by the pull down menu *Insert* or by activating the according button in the *Product Structure Tools* toolbar.

CATIA positions the inserted components according to their location regarding the coordinate systems in the Part structure. The four *Parts* can be seen in the *Specification Tree*. If nothing else is specified, CATIA labels the components in order of their insertion. The properties window allows a renaming of the elements.

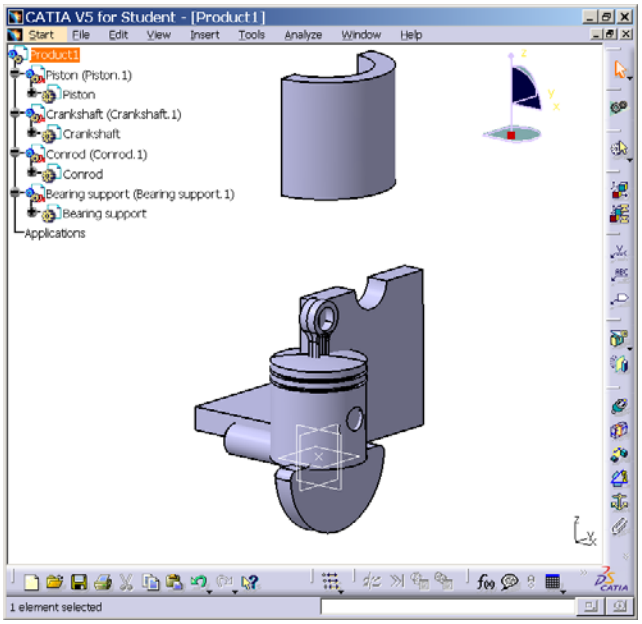


Figure 146: *Product* with the unconstrained but renamed *Parts*

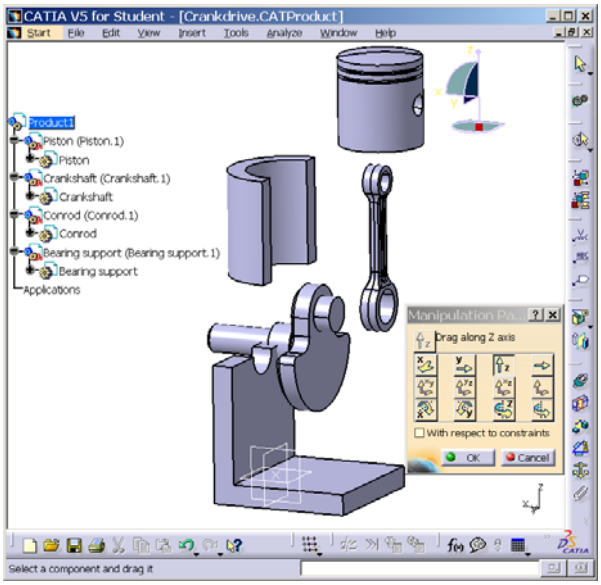


Figure 147: Roughly positioned assembly

For improving clarity, the components can be moved roughly to their final position with *Move / Manipulation.*

The precise positioning is done by defining *Constraints*.

First the *Bearing support* should be fixed in space as a reference. The next *Constraints* are applied to the *Crankshaft* which has to be positioned relative to the *Bearing block* using a *Coincidence Constraint* (axis of the *Crankshaft* and axis of the *Bearing block*).

After activating the feature *Coincidence Constraint,* CATIA recognizes the axis when clicking on the cylinder surfaces. The selected items are marked red.

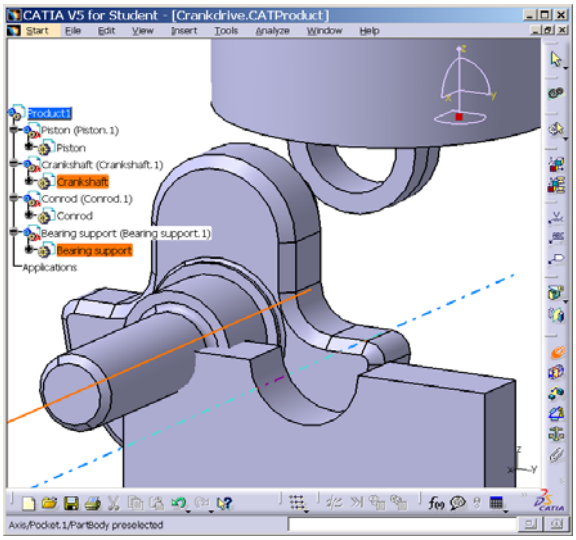


Figure 148: Using the *Coincidence Constraint* on two axes

By pressing *Update,* the *Crankshaft* is moved to the desired position, according to the constraints.
The *Conrod* has to be connected to the crankpin with a *Coincidence Constraint* and an *Offset Constraint.*

In the *Offset* input window the distance (*Offset*) between the two elements and the *Orientation* have to be set. As *Supporting Elements Planes, Points, Surfaces* etc. are suitable.
The degrees of freedom of the positioned *Conrod* can be viewed by choosing *Conrod.1 object / Component Degrees of Freedom* in the properties window.

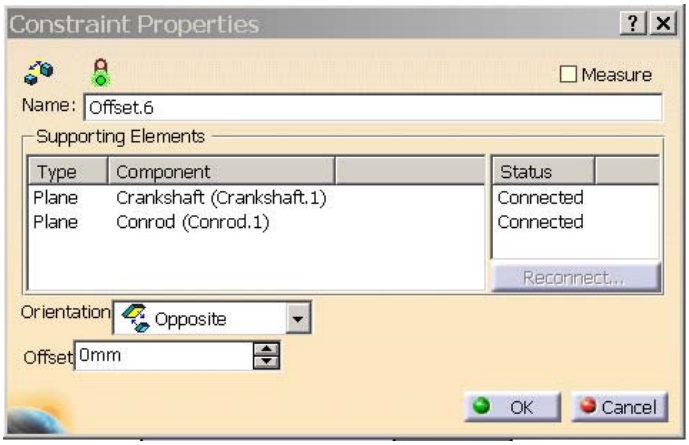


Figure 149: Dialog box of the *Offset Constraint*

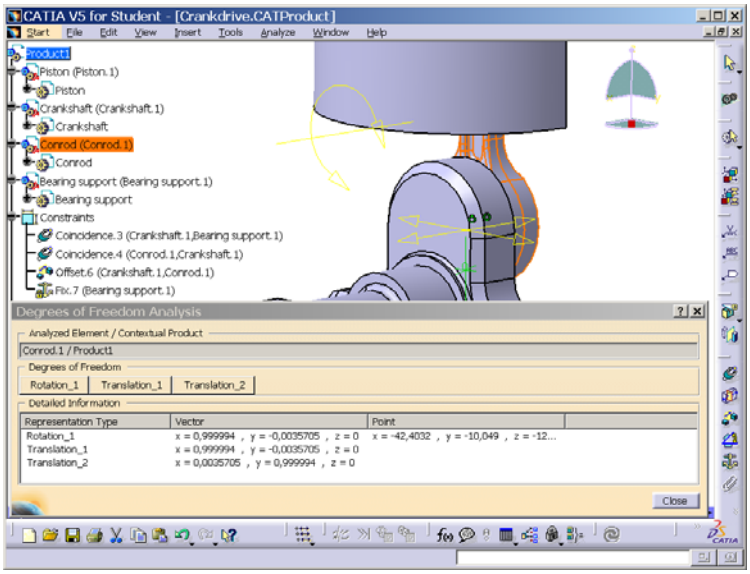


Figure 150: Analysis of the degrees of freedom

As the *Crankshaft* is still rotating around the crankpin axis and relocatable in direction of this axis, the *Conrod* has four degrees of freedom, which are displayed in the appropriate window. Those degrees of freedom are to be reduced.

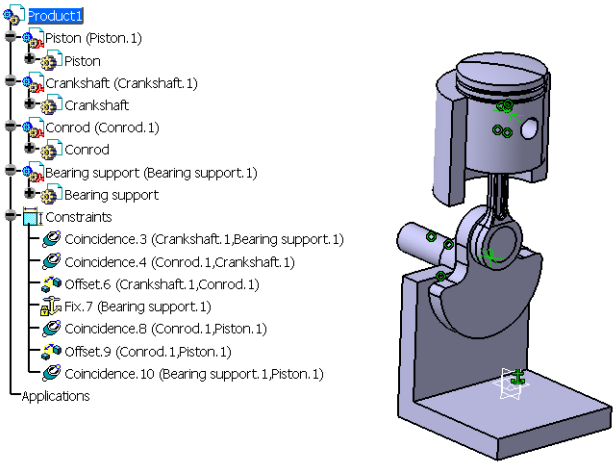


Figure 151: Complete assembly

The piston-pin bearing is connected with the upper *Conrod* bearing using a *Coincidence Constraint* and an *Offset Constraint*. The *Piston* itself is positioned in the *Cylinder* with a *Coincidence Constraint* of the two axes (*Piston* and *Cylinder*).
Figure 151 shows the completely constrained assembly. The *Crankshaft*, the *Conrod* and the *Piston* still have one degree of freedom each, which results in a movable crankdrive. The *Constraints* are displayed in the *Specification Tree* and in the design area; they can be switched to noshown space, if desired.

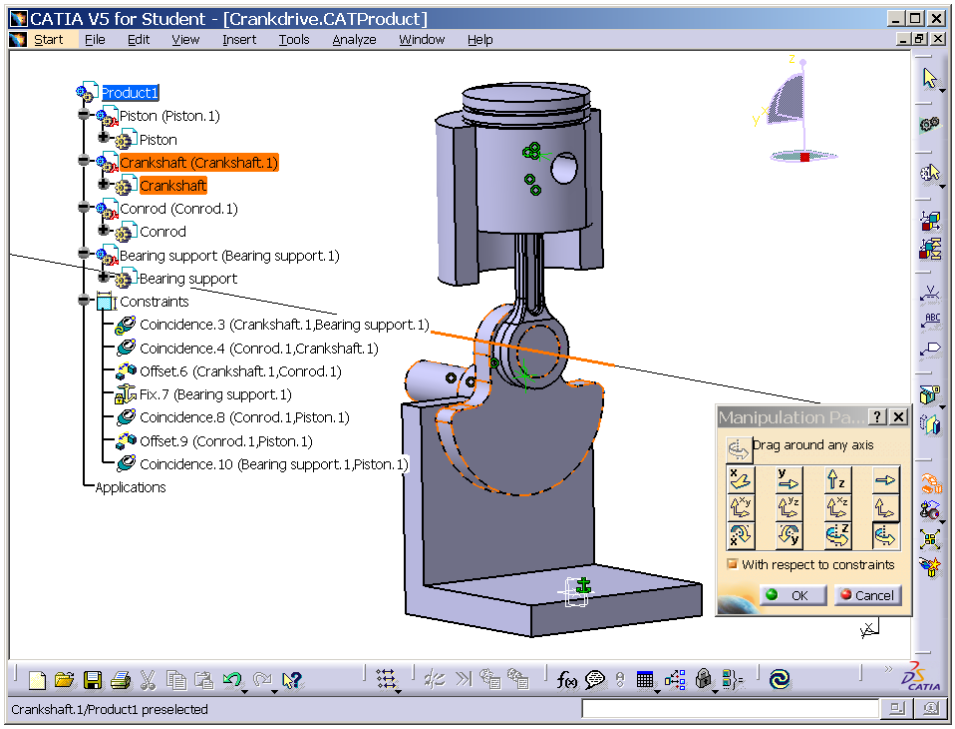


Figure 152: Rotating the *Crankdrive* using *Manipulation / With respect to constraints*

When activating the feature *With respect to constraints* in *Manipulation*, the assembly can be moved with respect to its constraints.

The feature *Explode* enables a quick generation of exploded views.

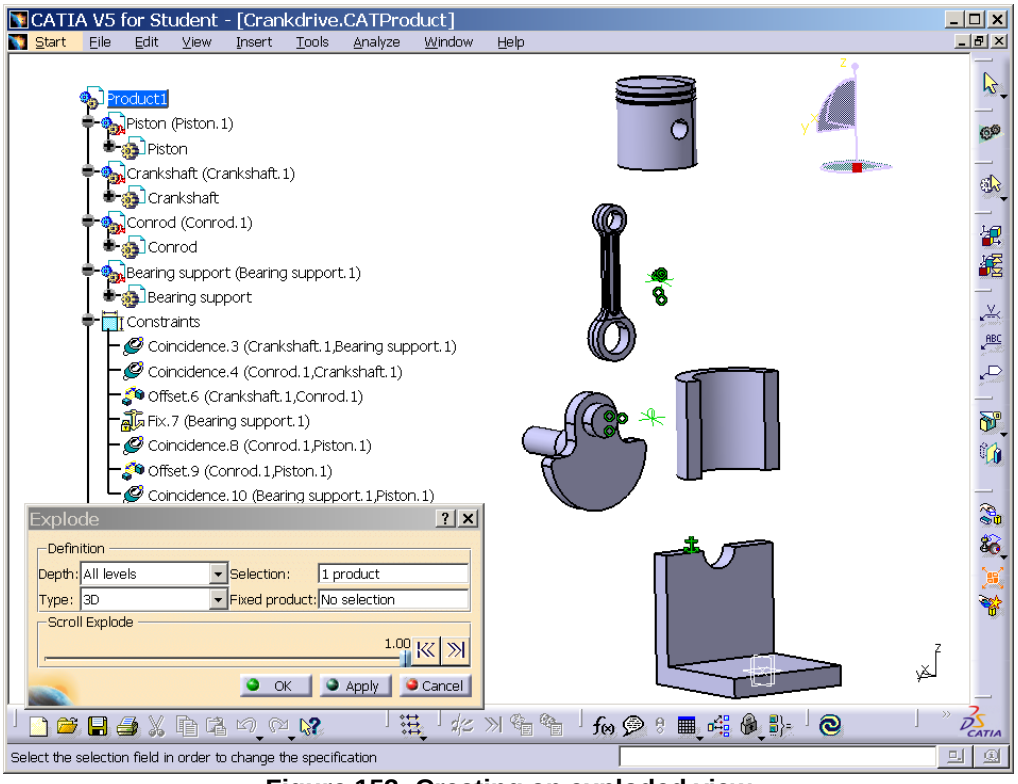


Figure 153: Creating an exploded view

Challenge: There is one error within this example which can be detected when carrying out a clash analysis.

Example 26: Clamping device

Intention: Create an assembly

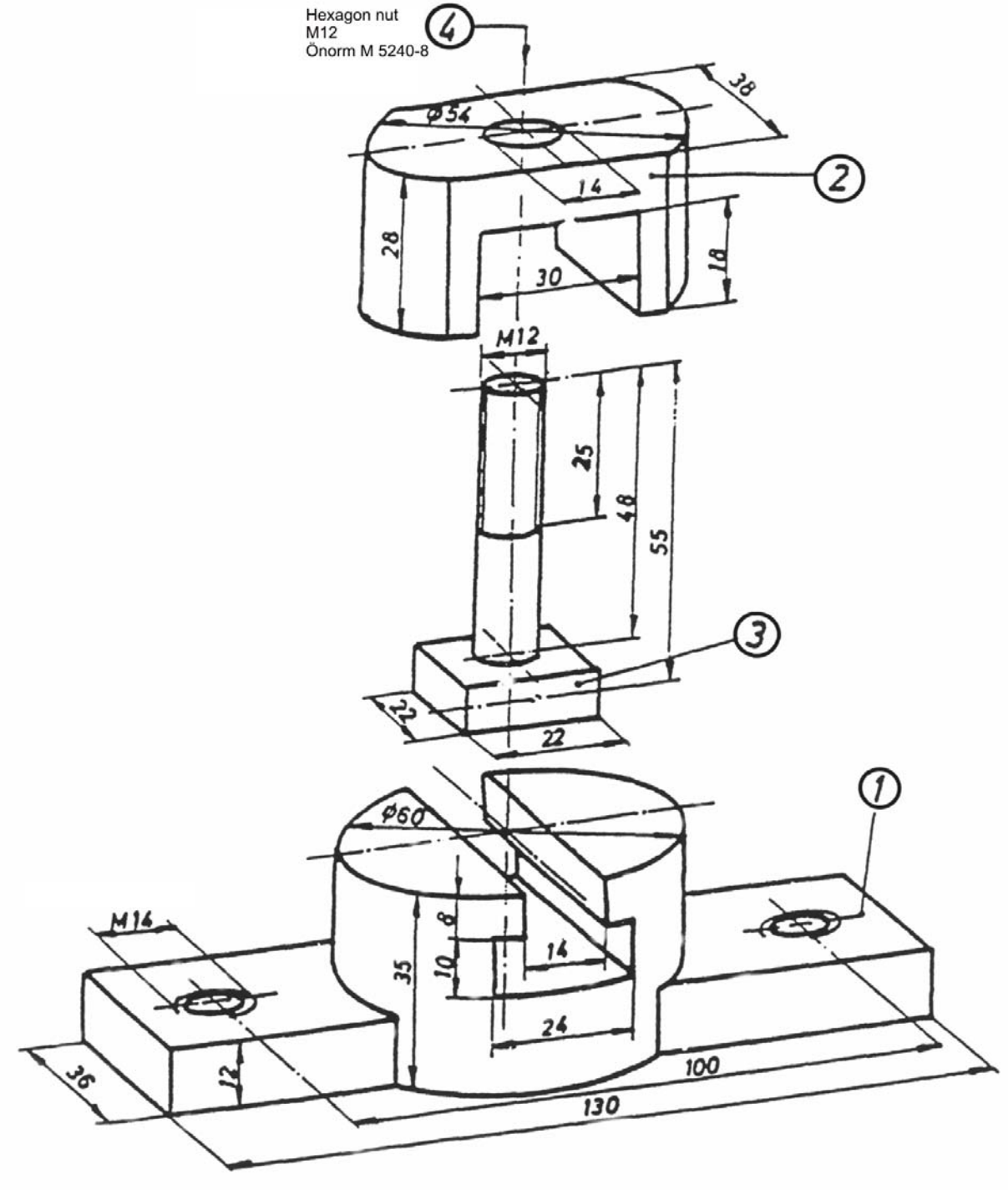


Figure 154: Clamping device

10 Excerpt of data management

CATIA V5 provides several other data formats apart from the standard CATIA files. This can be useful when exporting data to other CAD software. Different ways of data export are treated in the course of this chapter.
Data import can be carried out as well; the specific case of importing CATIA V4 data is an important topic which will be explained too.
The basic settings for data import and export can be adjusted in *Tools / Options / General / Compatibility*.

10.1 Exporting 3D data

Most of the 3D data can be transformed into the desired format by simply saving the element (*File / Save as)* with the according extension.

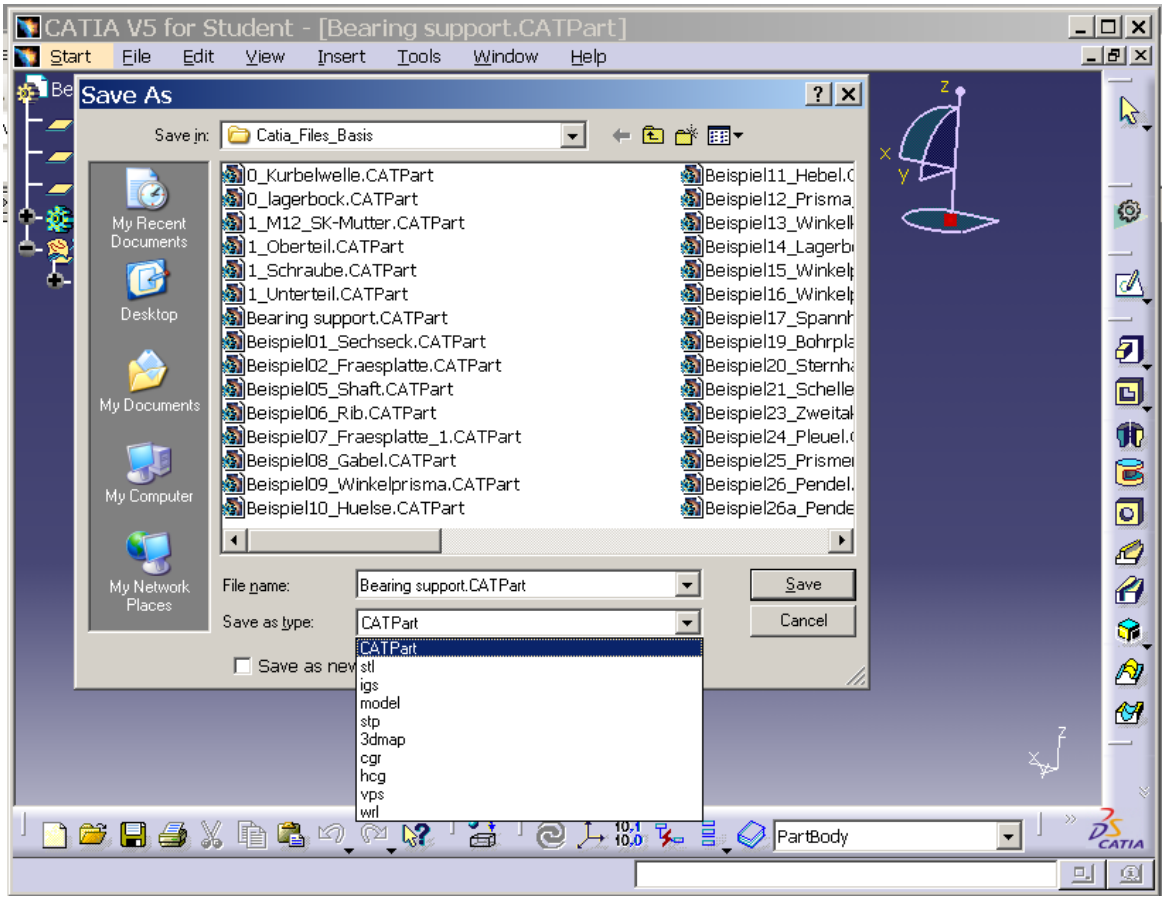


Figure 155: Saving options for a *Part*

The following listing describes some of the most frequently used formats for *Parts* and *Products*.

- *cgr* … The *cgr* format includes simplified geometry data and can be used as a memory efficient way to handle complex elements in CATIA, especially for DMU applications. The *cgr* files cannot be opened in CATIA by simply selecting *File / Open*, but they have to be inserted in a product using *Components / Existing Component*.

- *stp* … *Step* files of CATIA V5 *Products* still contain the *Product* structure. They are set up from volumes, points and surfaces. As with the other formats, the build-up of the parts can not be retrieved, i. e. the volumes can be viewed but not edited (The geometry information is without “history”.). The *Step* files can be opened via *File / Open.*
- *igs* … These files consist of points and surfaces. They can be opened similar to the *Step* files. The geometry information is without “history”.

10.2 Exporting 2D data

2D data such as drawings can be exported as well. Common formats are:

- *dxf*
- *dwg*

They are both vector graphics.

10.3 CATIA V4 data

Importing and exporting CATIA V4 data to and from CATIA V5 requires special methods.

Converting CATIA V5 data to CATIA V4:
To convert V5 data into data usable for V4, the operation *File / Save as* can be used. For *Parts* the format option *model* has to be used, for *Products* the appropriate format is *session*.

Converting CATIA V4 data to CATIA V5:
This conversion can be done in different ways.

A simple way is to open the V4 files using *File / Open*. This possibility, being the quickest one, has a drawback: The elements can be viewed, but not modified as the build-up information is enabled. The same characteristics apply to V4 elements that have been inserted in a *Product* via *Components / Existing Components*. Therefore V4 elements inserted like this are suitable for DMU analyses. An alternative way is to open V4 elements with *File / Open* and copy the specification tree (MASTER). The tree has to be inserted into an existing CATIA V5 *Part*.

If the V4 data is to be modified later on, it has to be converted using a special tool which can be accessed with the menu *Tools / Utility / Migrate V4 to V5*.
An alternative way is to open the V4 element with *File / Open* and copy the specification tree (MASTER). The tree has to be inserted into an existing CATIA V5 *Part*.

10.4 File administration

Save management:
The *Save Management* enables a selective saving of CATIA elements. This option is useful, especially when working with *Products*. The menu *File / Save Management* opens a dialog box. The *State* column shows whether an element is modified or not.

By marking an element and selecting *Save*, it is marked to be saved. Independent saves can be enabled by clicking the according button; this feature deactivates saves that are caused by other saves. After clicking OK, the preselcted items are saved automatically.

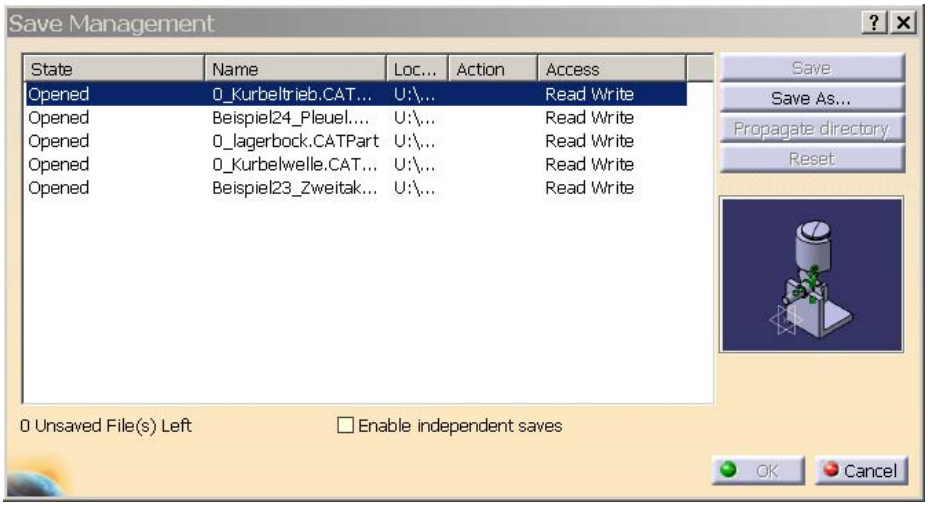


Figure 156: The *Save Management* window

Send To:
To transfer data to another directory or to other data media, the feature *File / Send To* can be used. By using this function, the file structure can be preserved, which means that the moved elements keep their relations.

Desk:
The *Desk* (*File / Desk*) graphically displays the links between the single elements (see page 86).

CATDUA:
CATIA provides a check and clean program for erroneous files named CATDUA. It can detect errors and, if possible, mend them. To activate it, the menu *Tools / Utility / CATDUAV5* can be used; alternatively, *CATDUA* can be opened by right-clicking the selected element on the *Desk.*

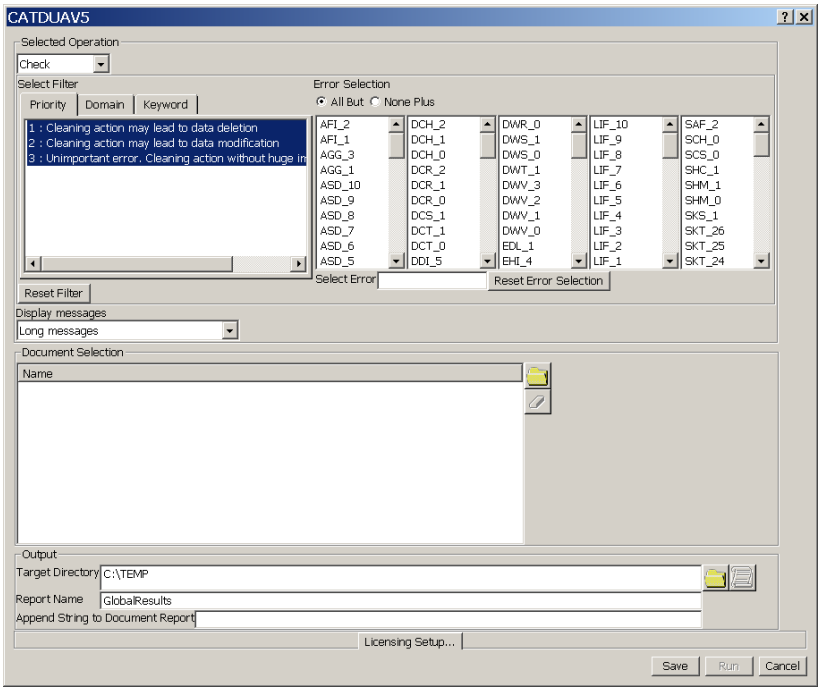


Figure 157: CATDUA

10.5 Publication

Making elements available to other users can be done by publicating them. When using the assembly design mode, this can be quite useful. To publicate elements, choose the menu *Tools / Publication.*

11 Creating drawings in the workbench *Drafting*

The *Drafting* mode in CATIA V5 enables the generation of 2D drawings. In principle all 2D views, sections, unfolded views, details etc. are connected with the 3D design elements. Additionally to deriving drawings from 3D data, CATIA allows the creation of independent 2D geometries.

The *Drafting* mode can be activated by either opening a new drawing (*File / New / Drawing*) or by selecting the workbench *Mechanical Design / Drafting* in the *Start* menu.

The definition window of the new drawing offers options for the *Standard*, the *Sheet Style* and *Size* of the drawing and the *Scale*.

The user interface of the *Drafting* mode is similar to the one in the 3D design mode. In the center the worksheet is positioned, the *Specification Tree* can be found on the left side, the toolbars are arranged around the worksheet.

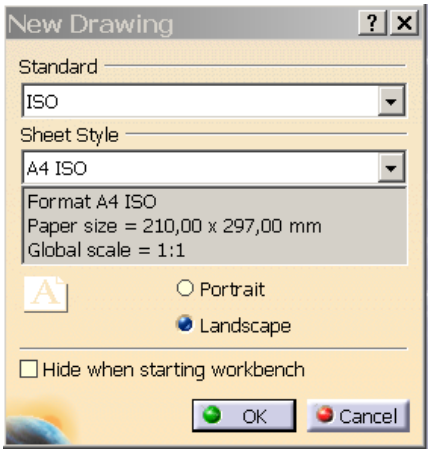


Figure 158: Definition box of the drawing

The manipulation features that can be carried out with the mouse buttons correspond to the ones in the 3D design mode. When pressing and holding the middle mouse button, the moving mode is active. After clicking the left mouse button once, the zooming feature is activated. The *Specification Tree* can be hidden by hitting the F3 button. If more than one sheet has been inserted in the drawing, it is possible to switch between the sheets by means of the bar at the top of the worksheet ('*Sheet.1*') or by selecting the sheet in the *Specification Tree*.

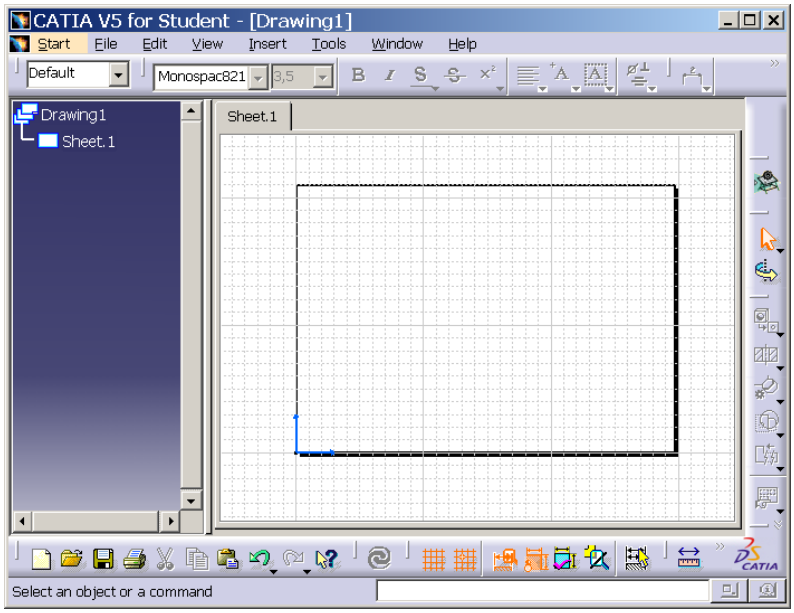
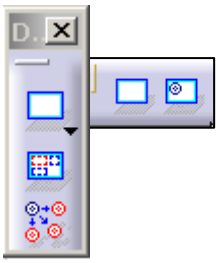


Figure 159: The *Drafting* workbench

To control the operations, the pull down menu *Insert* or the toolbars (they are set up in the menu *View / Toolbars*) can be used. An excerpt of the most important toolbars is shown subsequently.

11.1. Operations in the Drafting workbench

11.1.7 Drawing

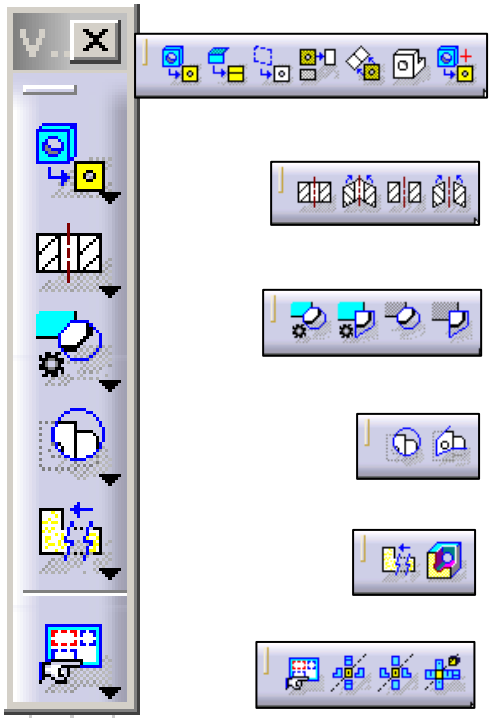


New Sheet, New Detail Sheet ... Creates a new sheet

New View ... Creates a new (empty) view

Instantiate 2D Component ... Creates a 2D component instance

11.1.8 Views



Create views: Different views can be selected:
Front View, Unfolded View, View from 3D, Projection View, Auxiliary View, Isometric View, Advanced Front View

Sections: Different directions of sectional drawings can be selected:
Offset Section View, Aligned Section View, Offset Section Cut, Aligned Section Cut

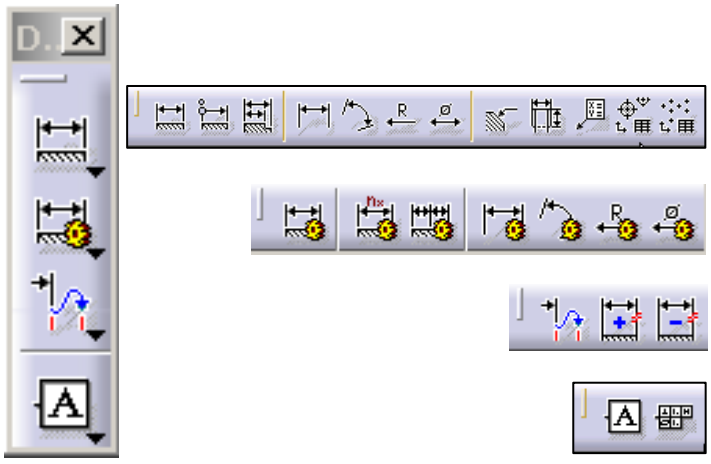
Details: Detail views can be created:
Detail View, Detail View Profile, Quick Detail View, Quick Detail View Profile

Clipping View: Create a Clipping View with a circle or a profile as callout:
Clipping View, Clipping View Profile

Broken View, Breakout View

View Creation Wizard: Different view options can be selected

11.1.9 Dimensioning



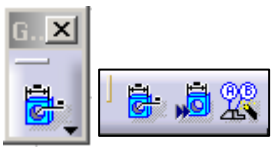
Dimension modes:
Distance, Angle, Radius, Chained Dimensions etc.

Technological Feature Dimensions

Reroute Dimensions, Create or *Remove Interruptions*

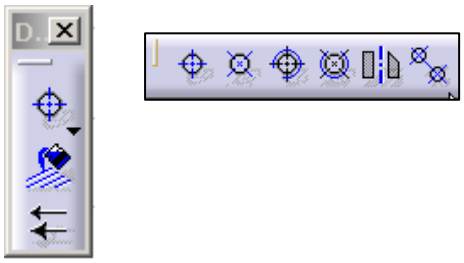
Datum Feature, Geometrical Tolerance

11.1.10 Generation



Automatically create dimensions and balloons:
Generate Dimensions
Generate Dimensions Step by Step
Generate Balloons

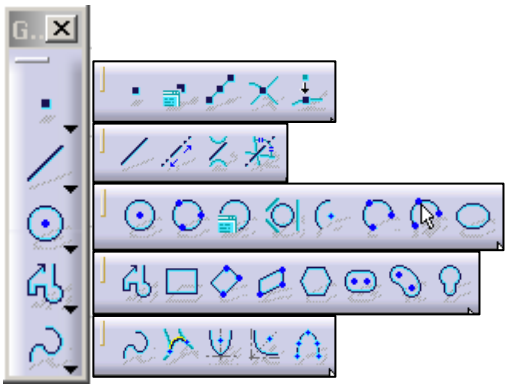
11.1.11 Dress-up



Create center lines, axis lines and threads in views or sections:
Center Line, Center Line with Reference, Thread, Thread with reference, Axis Line, Axis Line and Center Line
Area Fill ... Fill a selected contour

11.1.12 Geometry Creation

2D sketches can be created using the *Geometry Creation* features. They are to be used similar to the tools in the sketch mode.



Point creation features

Line creation features

Definition of circles and ellipses

Create profiles and polygons

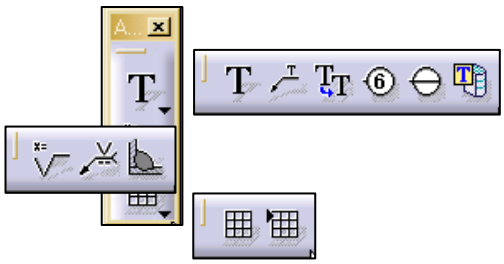
Create splines, connecting curves and conic sections

11.1.13 Geometry Modification

Corner, Chamfer, Trim, Break, Quick Trim, Close, Complement … Manipulation features for 2D geometry
Mirror, Symmetry, Translate, Rotate, Scale, Offset … Duplication and modification tools
Geometrical Constraint, Constraints Defined in Dialog Box, Contact Constraint … Define Constraints

11.1.14 Annotations

The toolbar *Annotations* can be used for inserting text, surface specifications, welding symbols and tables.



Text, Text with Leader, Text Replicate, Balloon, Datum Target, Text Template Placement

Roughness Symbol, Welding Symbol, Weld

Table, Table from CSV

11.2 The Properties Window

For creating a 2D drawing, the *Properties Windows* of the dimensions are important tools. The windows are activated by right-clicking the regarding dimension and selecting *Properties*. The *Properties Window* contains several settings, definitions and additional features.

Feature Properties ...	Allows renaming of the dimension and shows the creation date
Graphic ...	Graphical properties
Value ...	Value orientation, format, precision, fake dimension, etc.
Tolerance ...	Different options for tolerance values
Dimension Line ...	Graphical formatting of the dimension line

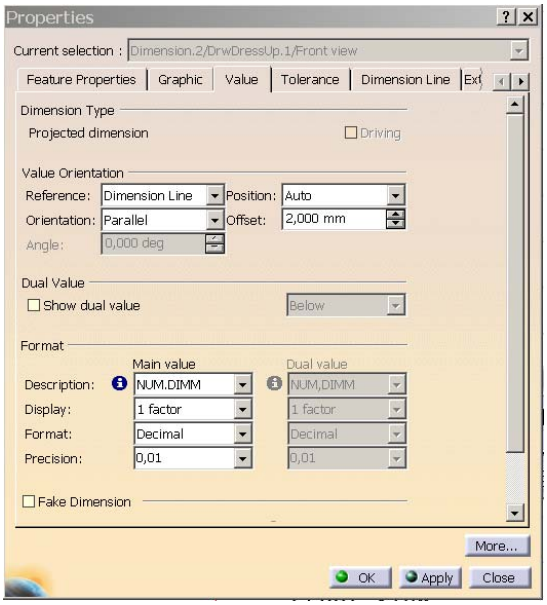


Figure 161: Properties window of a dimension

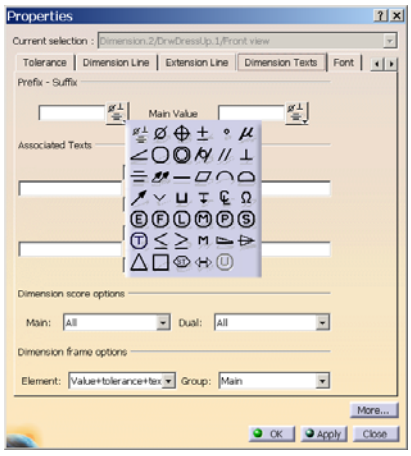
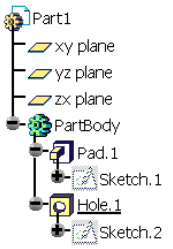


Figure 160: *Dimension Text* menu

Extension line ...	Graphical formatting of the dimension line
Dimension Text ...	Enables the input of text to the dimension
Font ...	Formats the font size, type and style
Text ...	Text settings

11.3. Basic steps for the creation of a dimensioned 2D drawing

By means of a simple example, the basic steps for creating a 2D drawing are shown. A drawing has to be deduced from a simple body. It should contain three basic views and a section view. Additionally, the drawing has to be provided with dimensions.



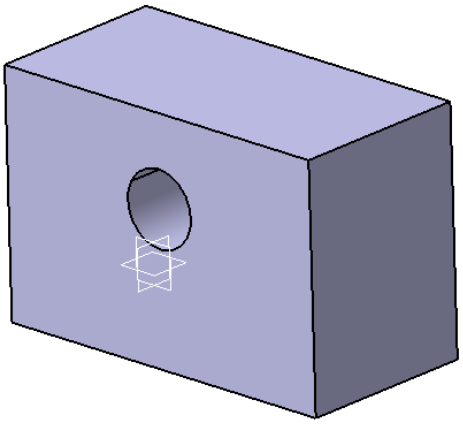


Figure 162: 3D *Part*

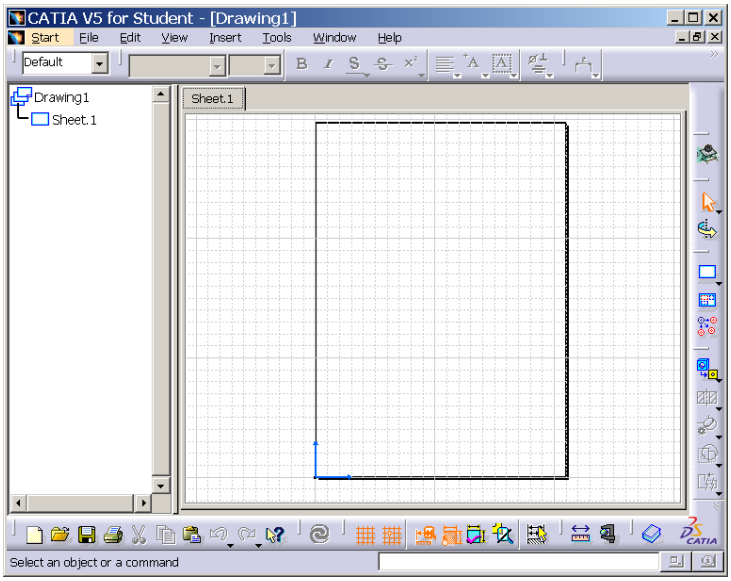


Figure 163: The *Drafting* workbench

After opening a new file *Drawing.1*, a DIN A4 sheet is selected in *Portrait* orientation.

The 3D *Part* is still open in background.

The feature *View Creation Wizard* can be used to select the desired views that should be displayed.

The *View Wizard* provides assistance for configuring the views.

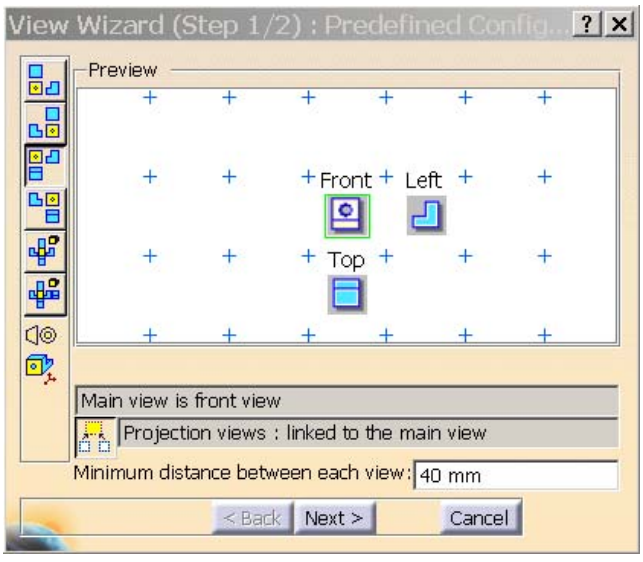


Figure 165: View Wizard

Single views can be created using the feature *Front View.*

After determining the desired views, a geometrical element has to be chosen in the according 3D editor. That's how to link a 2D *Drawing* and a 3D element. Right after clicking the geometrical element in 3D, CATIA switches back to *Drawing* mode and displays a preview of the views.

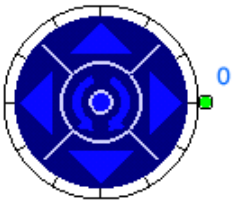


Figure 164: Control button

By means of the control button the views can be rotated. To finish the view creation, a click on the drawing sheet or on the middle of the control button settles the views.

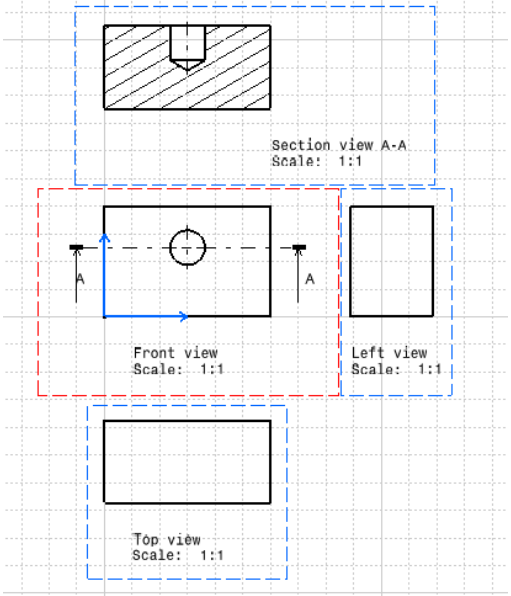
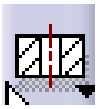


Figure 166: Different views of a *Pad* with *Hole*

Based on the defined views, a section view of the body can be created.



The feature *Offset Section View* enables the selection of the section profile. The section profile can be drawn by clicking the single points of the line with the left mouse button and finishing it is done by double clicking. The section is displayed according to the profile line.
The single views are bordered with a broken line.

All objects contained in the drawing are displayed in the *Specification Tree*. The object to be activated can be set active using *Activate View* (right mouse button).

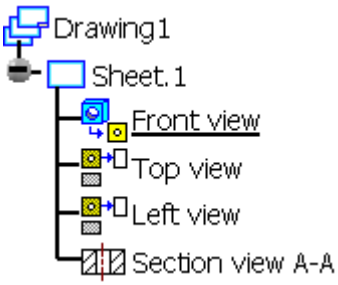


Figure 167: *Specification Tree* of a *Drawing*

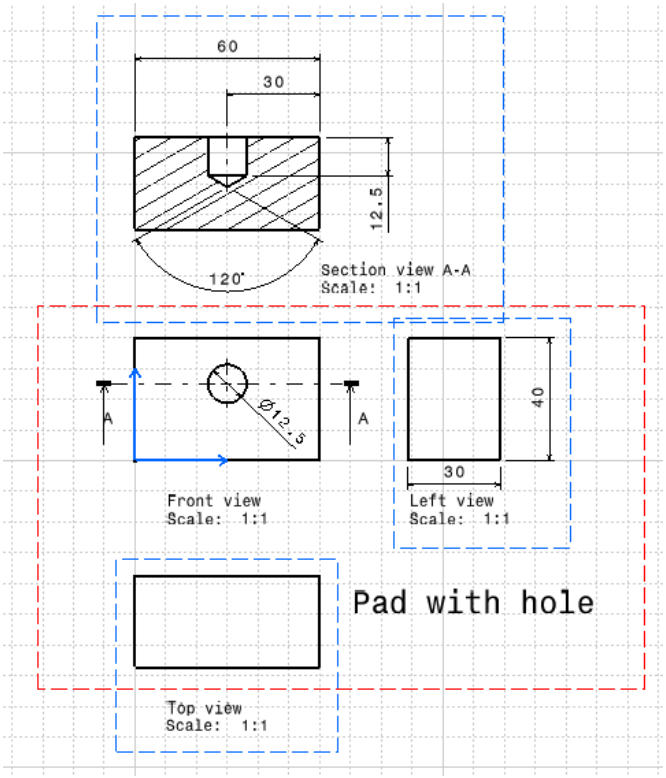


Figure 168: *Drawing* containing dimensions and text

Dimensioning is applied using the manifold features contained in the *Dimensioning* toolbar.

A text can be inserted by means of the toolbar *Annotations*.

☺ **Annotation:** All operations, such as *Dimensioning, Text,* inserted profiles etc. are added to the active object. Therefore, inconveniences may occur when objects are edited without being active.

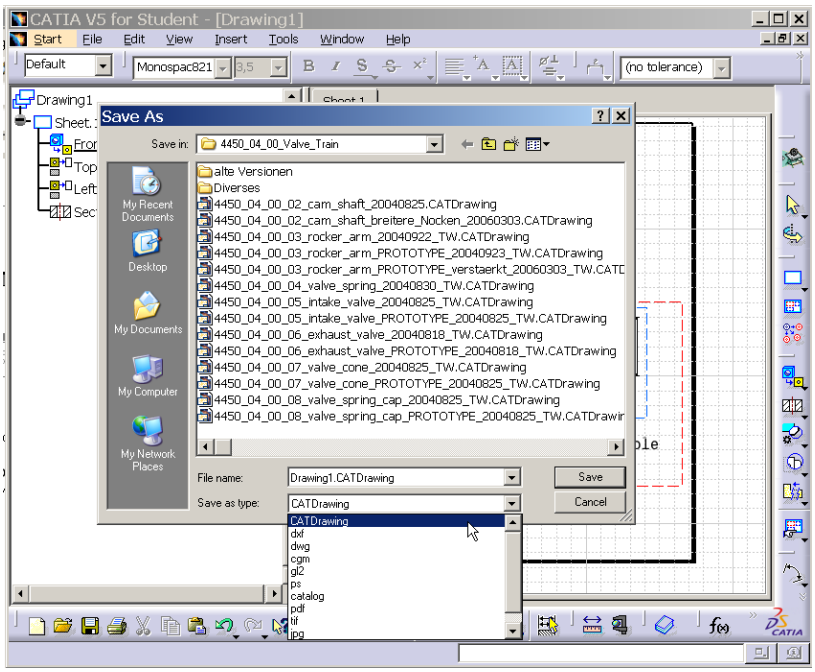


Figure 169: Saving options for a *Drawing*

Apart from the standard format *CATDrawing*, the drawing can be saved in several other exporting formats. Frequently used formats such as *dxf*- or *dwg*- can be used to import data to other CAD or CNC programs.

☺ **Annotation:** Objects like frames or title blocks can be inserted via the feature *File / Page Setup / Insert Background View.*

Additional helpful functions:

- Auto dimensioning (Function *Generation*)
- *View positioning* (in the context menue of the right mouse button)
- *Properties* in the context menue of the right mouse button
- Changing between working views and the background (in the pull down menue *Edit*)
- Toolbar *Geometry Creation*
- Toolbar *Text*

12 Create and use Parameters

Formula

Mathematical relations between parameters can be created and edited using the feature *Formula*. The corresponding dialog box can be opened with the menu *Tools / Formula* or the button *Formula* (toolbar *Knowledge*).

Figure 171: The *Knowledge* toolbar

The menu *Formula* lists all parameters used in the actual element. To enhance the overview, filters can be used to show only special parameter types. Figure 172 shows all *Length* parameters of a simple body.

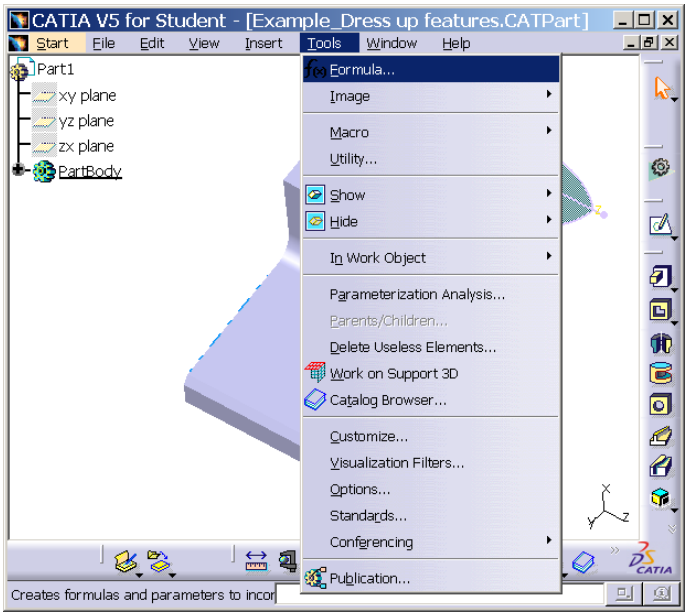


Figure 170: Starting the *Formula* editor

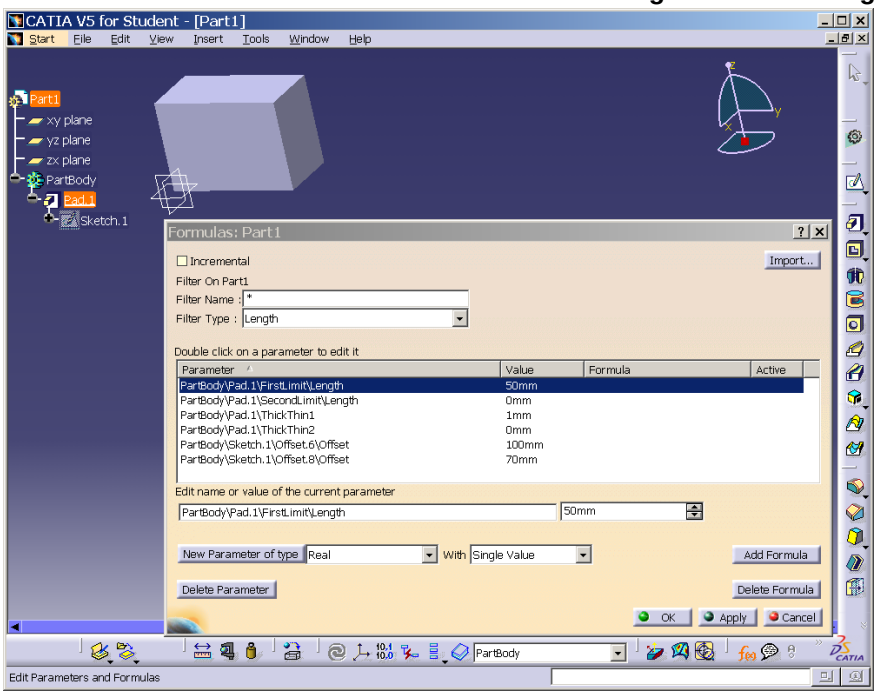


Figure 172: The *Formula* dialog box

When choosing *Import*, data from external programs can be integrated. New, user defined, parameters are inserted / deleted with the feautres *New Parameter* / *Delete Parameter*. The features *Add Formula* and *Delete Formula* enable the mathematical connection between the listed parameters. Activating *Add Formula* opens an additional window for the input of the desired formula. The parameter that has been marked afore is defined through the formula.

The solid body shown above measures 100 x 70 x 50 mm. The according parameters have been assigned automatically by CATIA.

width:	`PartBody\Pad.1\Length` ...	50 mm
length:	`PartBody\Sketch.1\Offset.6\Offset` ...	100 mm
height:	`PartBody\Sketch.1\Offset.8\Offset` ...	70 mm

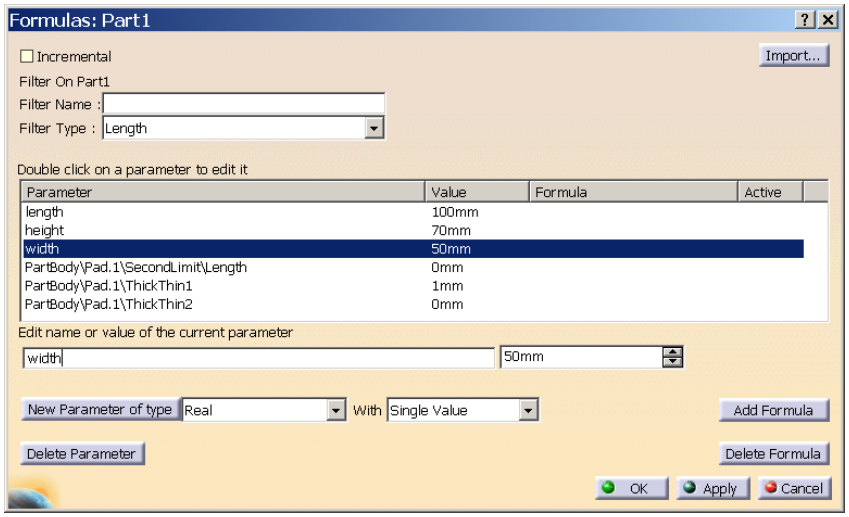


Figure 173: *Formula* editor with renamed *Parameters*

The parameters can be renamed by simply selecting them and modifying the name in the input line. Figure 173 shows the parameters that have been renamed from the default values to user defined names for width, length and height. Additionally, the length and the width should be linked in such a way that the *Pad* is three times as long as it is wide.

The parameter *length* has to be marked, the *Formula* editor opened and the parameter *width* is to be double clicked. Now the formula *length = width*3* can be set and OK can be picked. Therewith the desired relation is active and displayed in the *Formula* column.

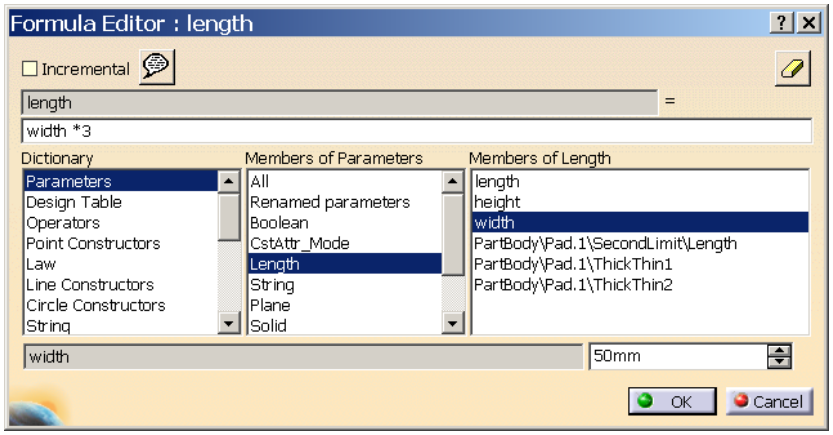


Figure 174: Entering a *Formula*

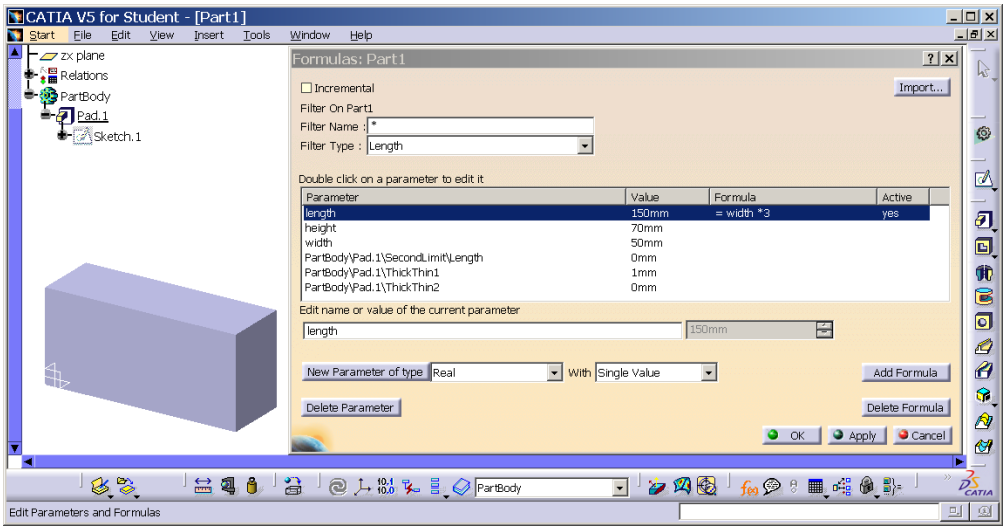


Figure 175: The *Formula* is displayed and set active

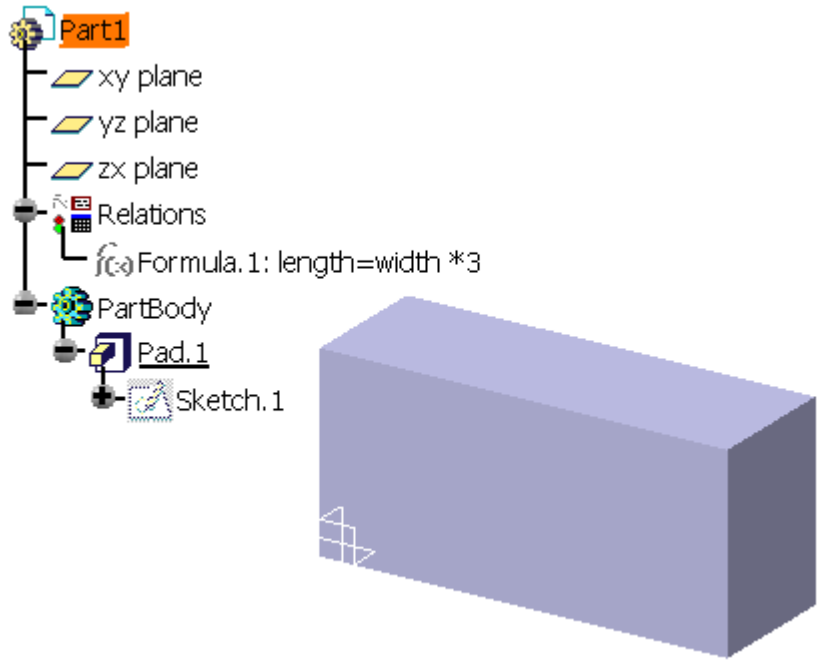


Figure 176: The *Specification Tree*, including the *Formula*

The dependency of the *length* from the *width* is displayed in the *Formula* window next to the value, and in the *Specification Tree* in the *Relations* branch. Double Clicking the relation in the *Specification Tree* opens the *Formula* editor. (To show the *Relations* branch in the tree, the according setting in *Tools / Options / Infrastructure / Part Infrastructure / Display* have to be adjusted.)